International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 09 Issue: 06 | Jun 2022 www.irjet.net p-ISSN: 2395-0072

Aerodynamic Analysis of Car body with Aerodynamic Devices to Improve Performance

V.V.R. MURTHY[1], D.K.V. TIRUMALA REDDY[2], D. VINAY PRAKASH[3], D. SATEESH[4], D. ESAKU[5]

[1]Associate Professor, Department of Mechanical Engineering, Nadimpalli Satyanarayana Raju Institute of Technology, Visakhapatnam, Andhra Pradesh, India
[2,3,4,5] Student, Mechanical Engineering, Nadimpalli Satyanarayana Raju Institute of Technology, Visakhapatnam, Andhra Pradesh, India

---

Abstract - *This research is about analysis of the effects of different Aerodynamic add on devices on the vehicle to reduce drag and make the vehicle fuel efficient. The 3D model is developed in ANSYS Space claim. Computational fluid dynamics (CFD) is performed to understand the effects of these add on devices. CFD is performed in ANSYS Fluent module. Drag Coefficient, lift coefficient, drag force and lift force are calculated and compared. The results are analyzed and it was observed that optimized body has better drag coefficient and lift coefficient which helps improving the fuel economy and stability of the car.*

Keywords: Aerodynamics, Drag Coefficient, Lift Coefficient, Computational fluid dynamics, Stream lined body.

1. INTRODUCTION

Aerodynamics plays crucial role in Automobile's performance. Initially, aerodynamics was used in racing to increase performance of the race cars to increase the race pace. But when fuel economy became a factor in road vehicles due to high prices of fuel, automobile manufacturers started to make changes to road cars by making them more streamlined and adding diffusers to reduce drag and improve fuel economy.

There will be different types of forces acting on a vehicle when it is moving. One of the forces that is acting against the flow of the vehicle is Drag Force. Reduction of this drag force helps in making a vehicle more fuel efficient and stable vehicle. The basic formula to calculate the overall drag is given by:

$$D = \frac{\rho}{2} C_d A V^2$$

Where,

C_d = Coefficient of Drag

A = Frontal area

V = Relative velocity of the object w.r.t. fluid medium

P = Density of air

So, our aim is to reduce the drag and lift forces acting on the car. We have analyzed 3 different models.

- Bluff model
- Streamlined model
- Streamlined model with diffuser.

And the results were noted and due to changes in the shape of the car body the drag reduction was observed which in return will increase the fuel efficiency of the car.

1.1 Computational Fluid Dynamics

CFD analysis consists of three main steps: Pre-Processing, Processing and Post-Processing. It is used to simulate fluid flow using computers with accurate results. We have used ANSYS Space Claim to design the Car models and ANSYS Fluent module to analyze the pressure contour, velocity contour, drag coefficient, lift coefficient, drag and lift forces of the models.

1.2 Concept of Streamlining

A body is stream lined when the air flow separation is low when compared to a bluff body whose air flow separation is high which causes a lot of pressure drag. A stream lined body has less pressure drag which in turn results in overall reduction in drag. A stream lined body is sleek and much easier to force such body through a fluid. So, we designed a car model with streamlined shape.

1.3 Diffuser

A diffuser, in an automobile, is a curved section of the car rear which improves the car's aerodynamic properties by improving the transition between the high-velocity airflow underneath the car and the much slower velocity in ambient atmosphere. The aft part of the car is where usually the diffuser is located. The diffuser helps in making the air flow at the exit is at the same pressure and same speed of the ambient.

2. Analysis

2.1 Bluff Model

First the Bluff model was designed and created using ANSYS Space Claim software.

a) Geometry

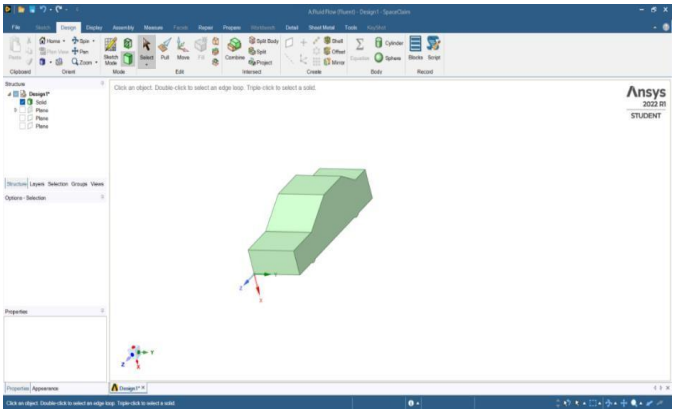

Fig -1: Bluff Model

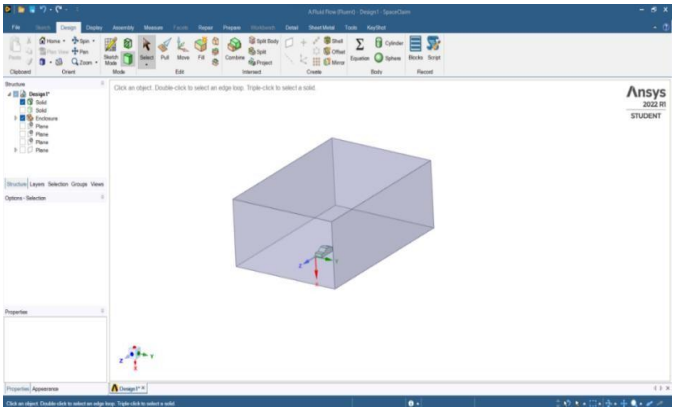

Fig -2: Enclosure created around Bluff Model

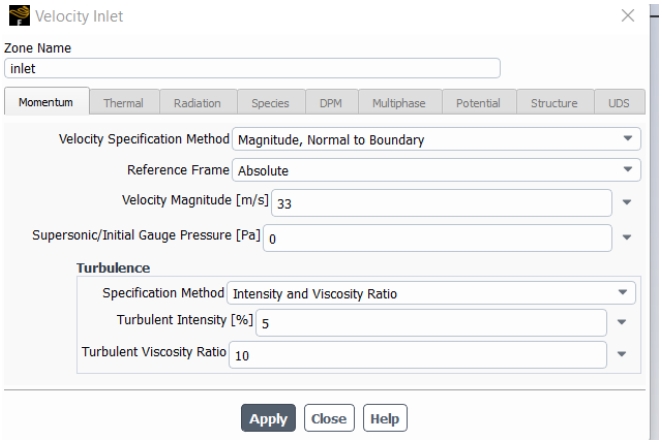

Fig -3: Inlet velocity given as 33 m/s

2.2 Streamlined Model

a) Geometry

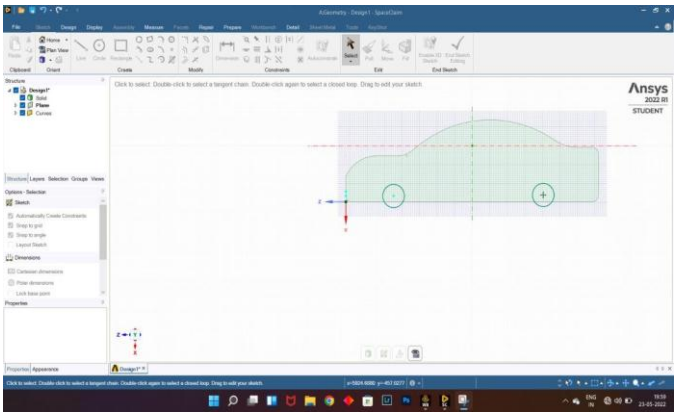

Fig -4: 2D view of Streamlined Model

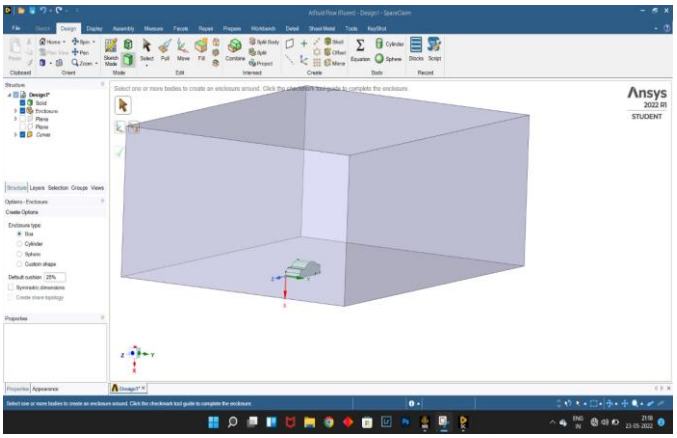

Fig -5: Enclosure Created around Streamlined model

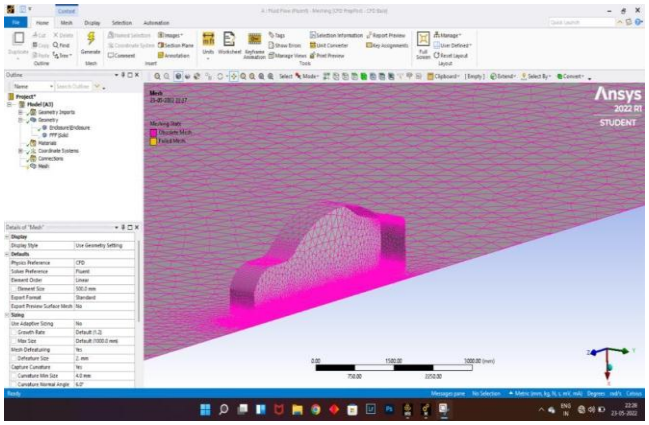

Fig -6: Sectional view of car model

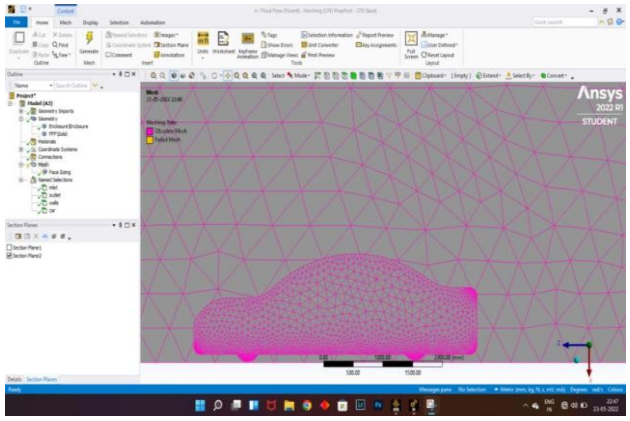

Fig -7: Extra meshing at rounded corners for better results

2.3 Streamlined Model with Diffuser

a) Geometry

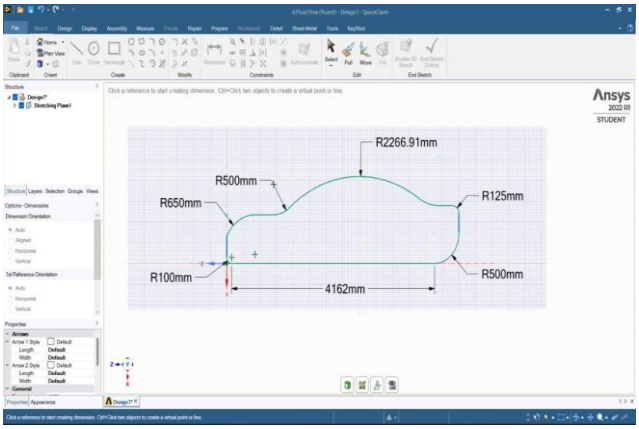

Fig -8: Streamlined Model with Diffuser

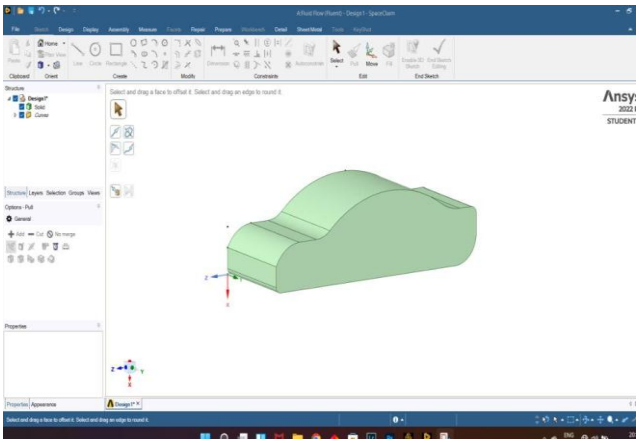

Fig -9: 3D Model of Streamlined model with Diffuser

3. CFD RESULTS

BLUFF MODEL

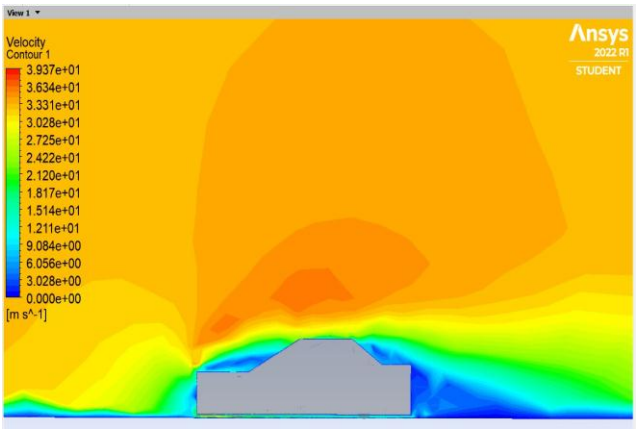

Fig -10: Velocity Contour of Bluff model

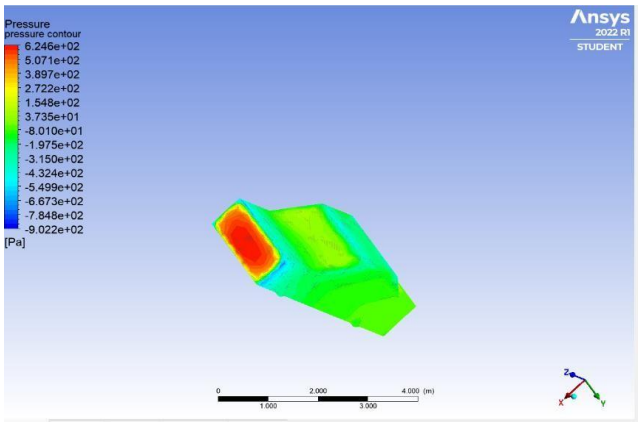

Fig -11: Pressure Contour of Bluff model

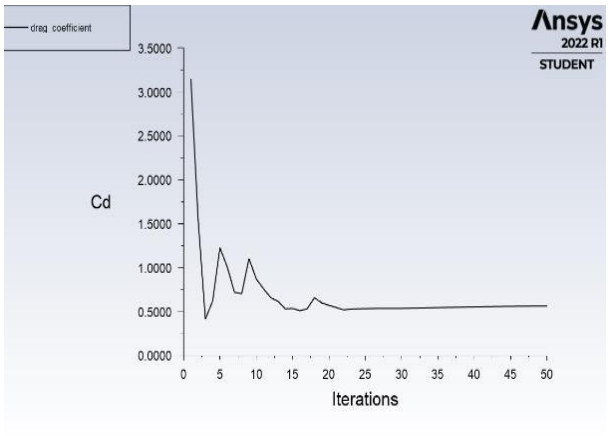

Fig -12: Cd plot

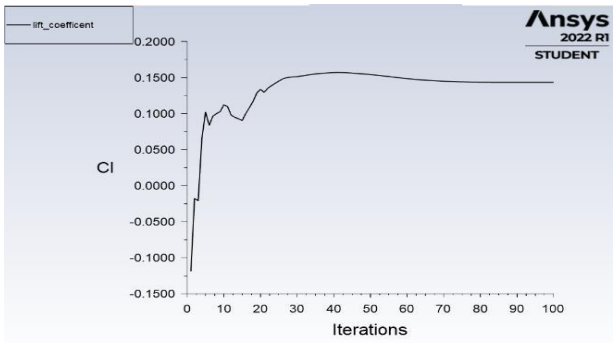

Fig -13: Cl plot

STREAMLINED MODEL

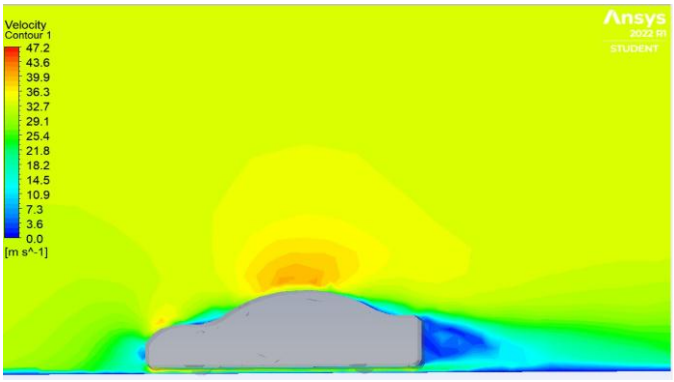

Fig -14: Velocity Contour of Streamlined Model

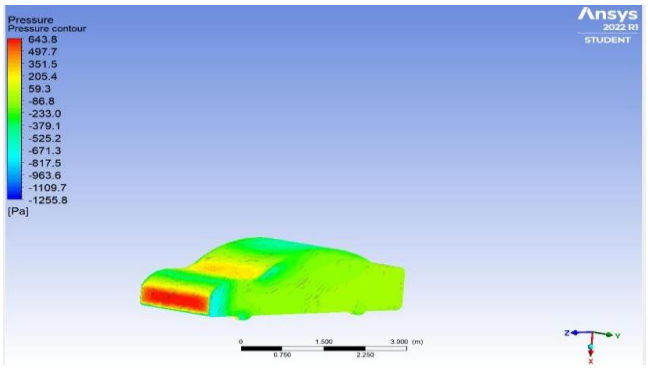

Fig -15: Pressure Contour of Streamlined Model

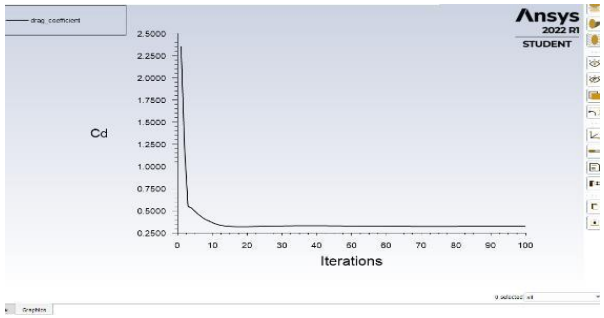

Fig -16: Cd Plot of Streamlined Model

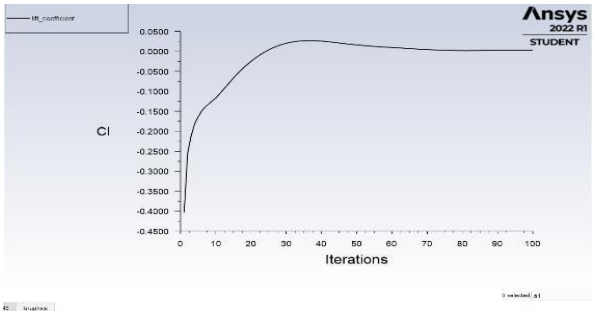

Fig -17: Cl Plot of Streamlined Model

STREAMLINED MODEL WITH DIFFUSER

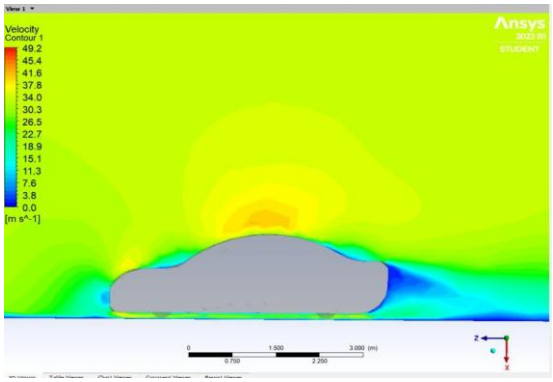

Fig -18: Velocity Contour of Streamlined Model with diffuser

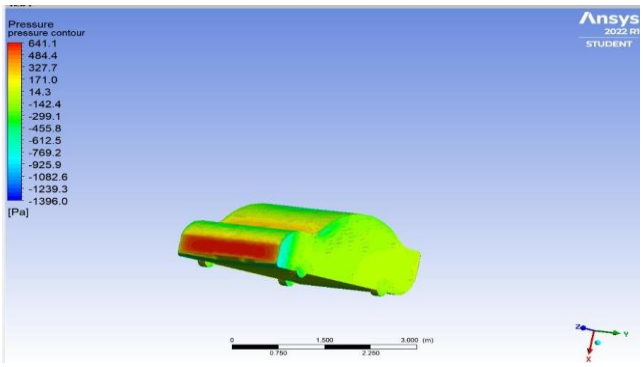

Fig -19: Pressure Contour of Streamlined Model with diffuser

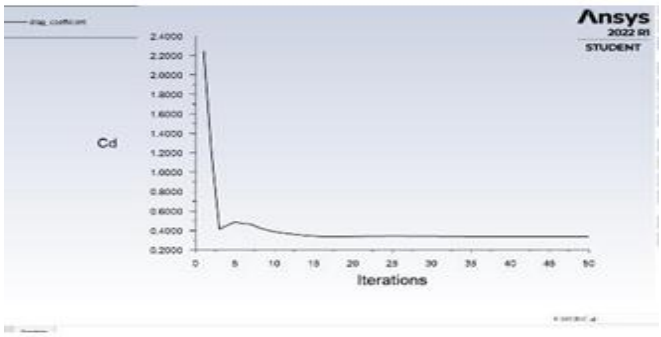

Fig -20: Cd Plot of Streamlined Model with diffuser

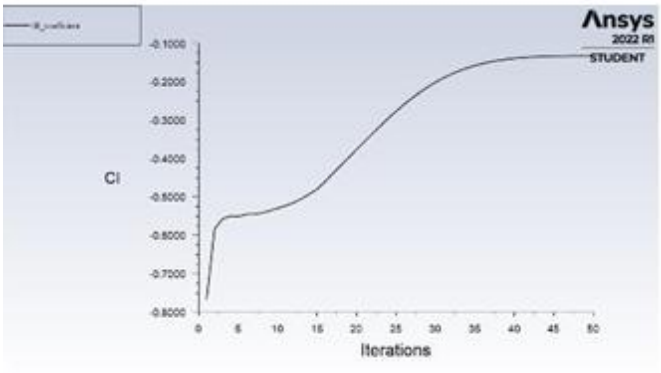

Fig -21: Cl Plot of Streamlined Model with diffuser

Table-1 contains the values of Coefficient of drag (Cd), Coefficient of lift (Cl), drag force and Lift Force of all three models.

	Bluff Model	**Streamlined Model**	**Streamlined Model with Diffuser**
Cd	0.5361	0.3242	0.3369
Cl	0.1540	0.0500	0.1000
Drag	1072.98(N)	612.07(N)	634.719(N)
Lift	270.41(N)	4.95(N)	2.615(N)

Table -1

From the above information on the basis of Cd, Cl, Lift and Drag Force the Stream lined model's drag and lift got significantly reduced when compared to bluff model. And the stream lined model with diffuser's lift got reduced slightly which in return increases the stability of the car at high speeds. So, the optimized Shape of stream lined model and diffuser offers better and lesser drag when compared to the other models. This helps in Increase of Fuel economy and also the highspeed stability of the car.

CONCLUSIONS

From the above study we arrive at the following:

Aerodynamics is an important factor to consider while manufacturing a vehicle. Here, in this paper, we created a Stream lined model and analyzed the aerodynamic parameters of the car including the drag produced during high performance. From the analysis performed it is observed that the car with streamlined body's drag is low but there is small amount of lift is generated. But the diffuser model's drag is low and also creates small amount of downforce which increase the grip of the car and provides good traction. This analysis shows the difference between a car with no Aerodynamic devices and car equipped with Aerodynamic devices. The Drag and lift are improved significantly and result will be better fuel efficiency and better highspeed stability.

REFERENCES

[1] Koike, M., Tsunehisa, N., and Naoki, H., Research on Aerodynamic Drag Reduction by Vortex Generators, Mitsubishi Motors Tech. Rev., vol. 16, pp. 11–16, 2004.

[2] Singh, S.N., Rai, L., Puri, P., and Bhatnagar, A., Effect of Moving Surface on the Aerodynamic Drag of Road Vehicles, Proc. Inst. Mech. Eng., Part D, (J. Automob. Eng.), vol. 219, pp. 127–134, 2005.

[3] Drag Force and Drag Coefficient, Sadraey M., Aircraft Performance Analysis, VDM Verlag Dr. Müller, 2009.

[4] Numerical Study on Aerodynamic Drag Reduction of Racing Cars by S.M. Rakibul Hassan, Toukir Islam, Mohammad Ali, Md. Quamrul Islam, ICME 2013.

[5] Methods of Reducing Vehicle Aerodynamic Drag by Upendra S. Rohatgi, Presented at the ASME 2012 Summer Heat Transfer Conference, Puerto Rico, USA.