iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 9 **Issue:** VIII **Month of publication:** August 2021

DOI: https://doi.org/10.22214/ijraset.2021.37853

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 VIII Aug 2021- Available at www.ijraset.com

Reduction of Aerodynamic Drag of Heavy Vehicles using CFD

Priyank Kothari[1], Elton Dias[2], Soumya Chaphale[3]

***Abstract:** Aerodynamic drag is the force that opposes an object's motion. When a vehicle no matter the size, is designed to allow air to move fluidly over its body, aerodynamic drag will have less of an impact on its performance and fuel economy. Heavy trucks burn a significant amount of fuel as to overcome the air resistance. More than 50% of an 18-wheeler's fuel is spent reducing aerodynamic drag on the highways.*

***Keywords:** Aerodynamics, Heavy vehicles, ANSYS, Aerodynamic Drag, Fuel efficiency.*

I. INTRODUCTION

A. Overview

In the recent years, demand for the fuel consumption of vehicles within the automotive industry is increasing. Increased fuel price and development of more fuel-efficient vehicles have raised this issue. Heavy commercial vehicles in comparison to other ground vehicles, due to high aerodynamic drag, have low fuel efficiency. Therefore, it is necessary to find a way to make these more fuel efficient.

B. Motivation

Recent research about fuel reduction technologies for trucks show that aerodynamic improvement is one of the most important technologies when it comes to fuel saving. A large commercial vehicle travelling at 100 km/h consumes about 50% of the total fuel to provide the power required to overcome the aerodynamic drag. It has been investigated that on average a heavy commercial vehicle's annual mileage varies between 130000 km and 160000 km. Due to such a high mileage, any reduction of aerodynamic drag will result in significant fuel savings and reductions in greenhouse gas emission. This also relates to high levels of pollution (CO2), from the burning of fossil fuels. Their results demonstrate that fuel economy improvement could be achieved from less than 1% to almost 9% of an annual mileage.

C. Background

Nowadays, modern trucks are equipped with a range of drag reducing mechanisms. A lot of numerical and experimental studies about the heavy vehicle aerodynamics have been carried out. Most of the research today is done on reducing drag on newly designed trucks. As a result, little work is done on current designs. Due to the high number of trucks already on the road today, as well as the fact that many of these older designs are still sold, it is imperative to find ways to reduce the drag on these designs.

II. THEORY

Most of the designs which are currently present on the roads are not designed taking the aerodynamics into consideration therefore wasting a lot of energy to overcome the drag. So, in this paper a model is being modified to make it more efficient. It has been investigated that on average a heavy commercial vehicle's annual mileage varies between 130000 km and 160000 km. Due to such a high mileage, any reduction of aerodynamic drag will result in significant fuel savings and reductions in greenhouse gas emission. In a research, it is found that over 1.3 trillion litters of petrol and diesel is consumed by road vehicles. This also relates to high levels of pollution (CO2), from the burning of fossil fuels. Increased fuel price and development of more fuel-efficient vehicles have raised this issue. Heavy commercial vehicles in comparison to other ground vehicles, due to high aerodynamic drag, have low fuel efficiency.

III. PROPOSED SYSTEMS

A. Addition of Deflector

The most popular supplementary part is deflector that is a flat and formed plate fixed on the top of cab at different angles. It is added to the simple box model and then it is meshed. Finally, current flow passes over it to evaluate the effect of this set on aerodynamic drag reduction.

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VIII Aug 2021- Available at www.ijraset.com

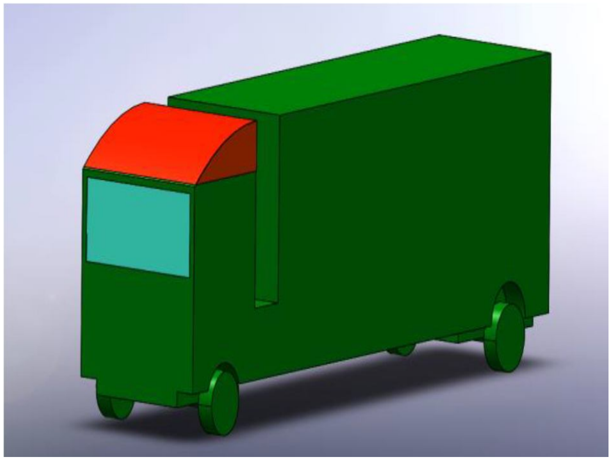


Fig – 1: Deflector

B. Traps

Base flaps are simple trapezoidal plates installed at the bottom of container and converge together to postpone the separation and regulate the turbulence. It is known from literature that the effect of base flaps depends on Reynolds number and the relations are so complicated.

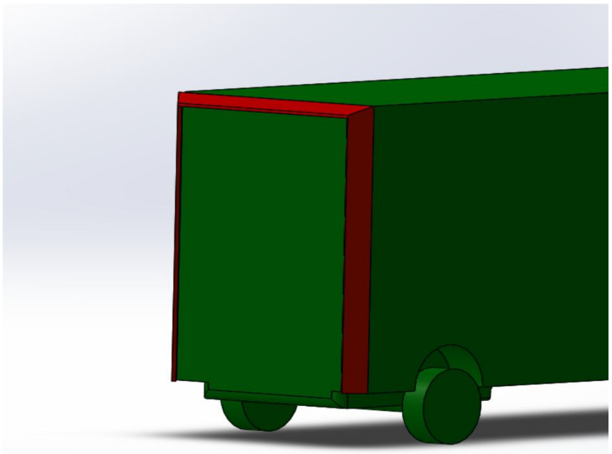


Fig – 2: Traps

C. Cab Vanes

Cab Vanes are the front body curvatures to reduce the frontal area thereby decreasing the drag resistance and deflecting the streamlines away from the truck.

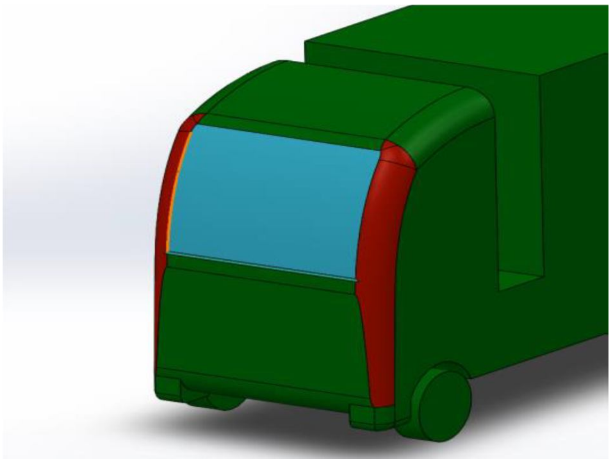


Fig – 3: Cab Vanes

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 VIII Aug 2021- Available at www.ijraset.com

D. Front Splitter

It is used for the smooth flow of air under the vehicle so that it may not be deviated or obstructed by various devices under the vehicle.

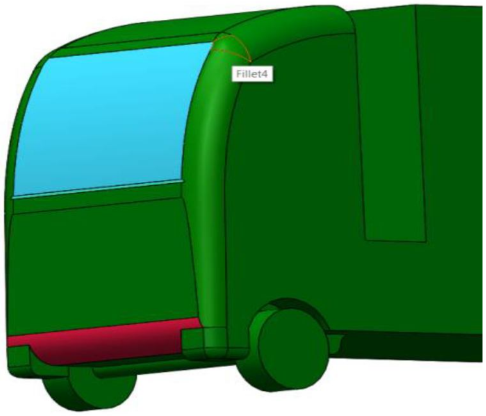


Fig – 4: Front Splitter

Several attachments or modifications in addition to the above-mentioned parts helps in reducing the drag for e.g., side panels, front fairings, underside panels, moreover the gap between the tractor and the trailer can be decreased to reduce the aerodynamic resistance of the vehicle.

IV. DESIGN

An enclosure was made to make a wind tunnel like system, consisting of inlet and outlet for depicting the actual flow of air on the roads.

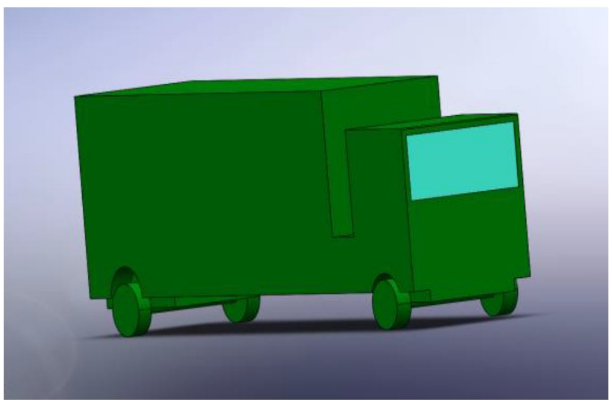


Fig – 5: Basic Model

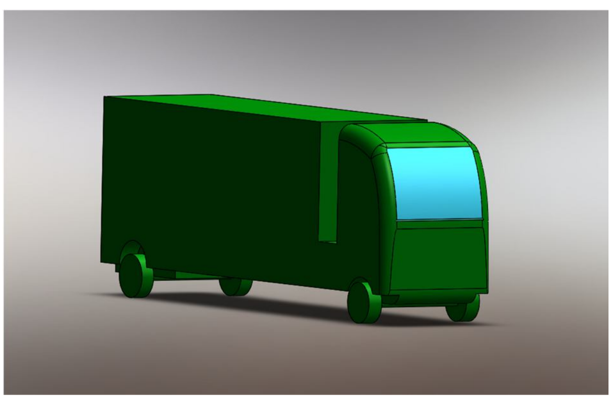


Fig – 6: Modified Model

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 VIII Aug 2021- Available at www.ijraset.com

V. ANALYSIS

At first, a truck is modelled as two boxes by using Solidworks and then the meshing is performed, and boundary conditions are defined. The vehicle is assumed to have speed of 27 m/s at 25 °C, at pressure of 105 Pa. The vehicle model was imported to the ANSYS Fluent environment**.** A mesh was being created and is shown in the figure below.

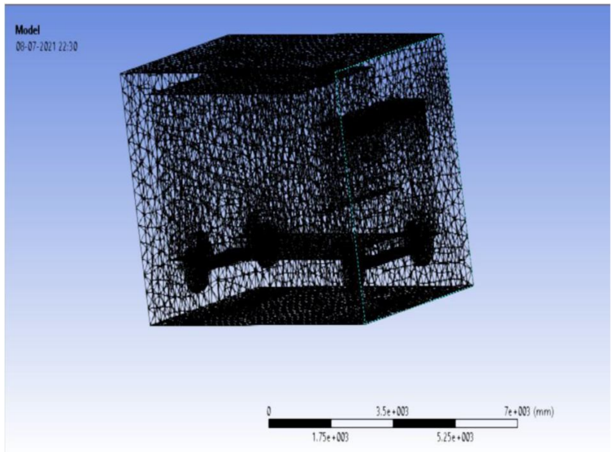


Fig – 7: Meshed Basic Model

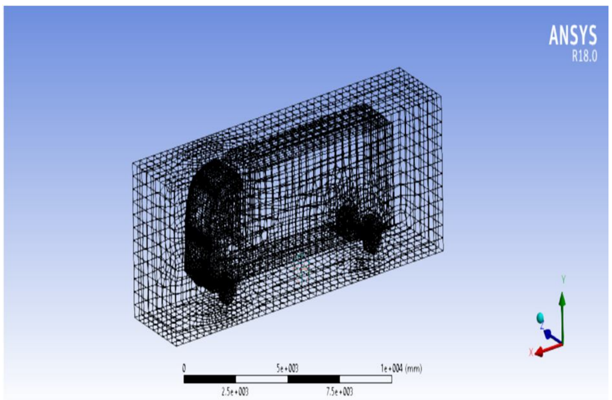


Fig – 8: Meshed Modified Model

After meshing, the inlet and outlet conditions are being set, the solver which is used for the analysis is K-epsilon. The total aerodynamic drag force on the body was calculated. The area were the maximum turbulence and pressure was studied and the design was being modified to minimize the drag.

A. Results Obtained on the Basic Model

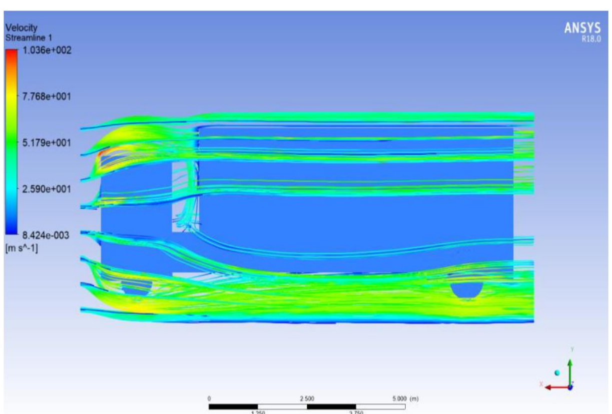


Fig – 9: Velocity streamline (basic model)

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 VIII Aug 2021- Available at www.ijraset.com

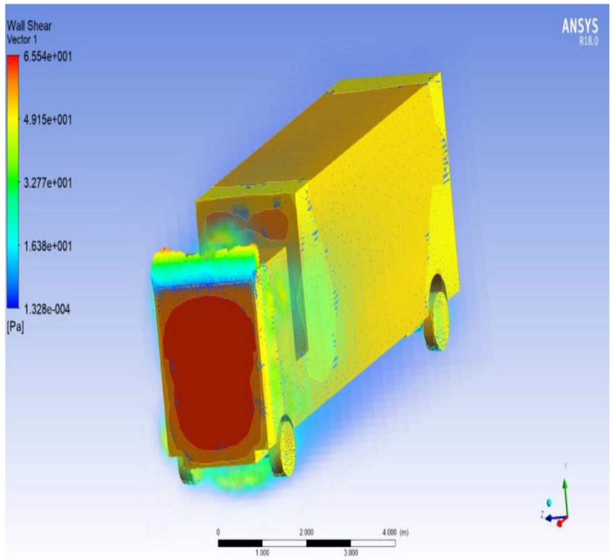


Fig – 10: Pressure contour (basic model)

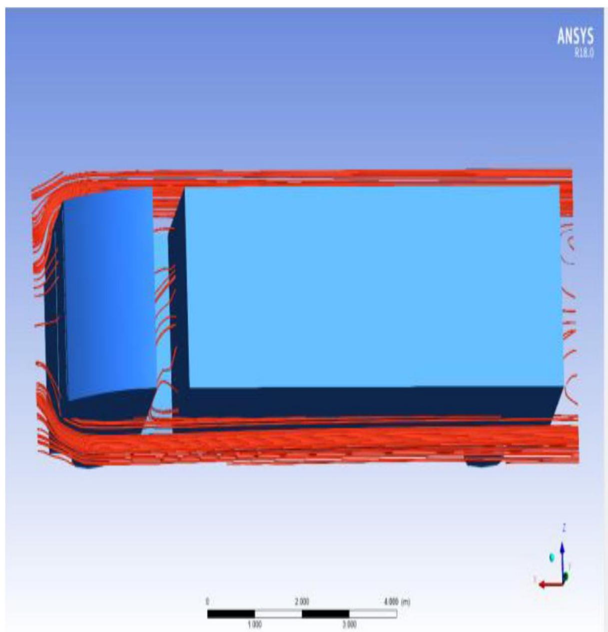


Fig – 11: Flow without side vanes

The total aerodynamic force obtained, and the coefficient of drag is shown in the following figure.

Forces (N)	
Pressure	40573.266
Viscous	477.76128
Total	41051.027
Coefficients	
Pressure	8.9943
Viscous	0.1059
Total	9.1003

Table -1: Drag force and Coefficient (Basic Model)

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VIII Aug 2021- Available at www.ijraset.com

B. Results Obtained on the Modified Model

Most of the drag force on the truck is observed to be on the frontal area, moreover the container is also directly subjected to the incoming air and the increasing the drag. Adding a deflector to the truck, above the truck's cabin will not only reduce the area perpendicular to the air flow but also will guide air away from the truck which in turn reduce the friction overcome by the body.

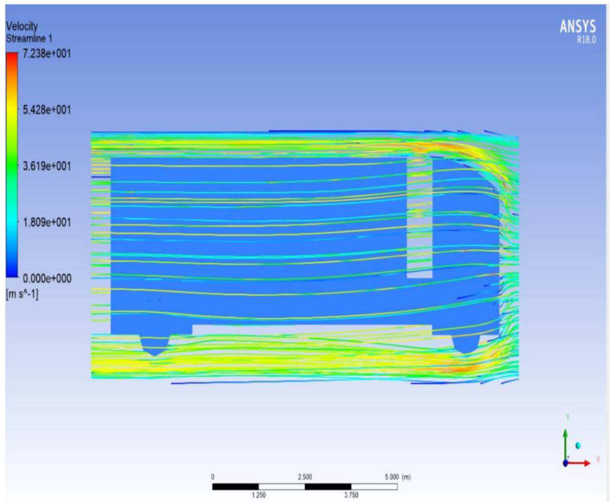


Fig – 12: Velocity Streamline (with deflector)

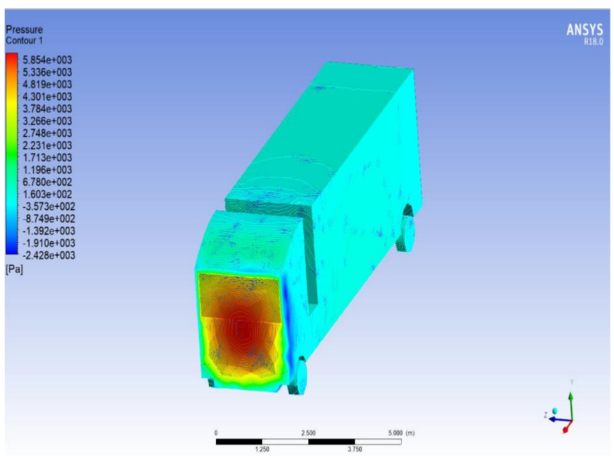


Fig – 13: Pressure contour (with deflector)

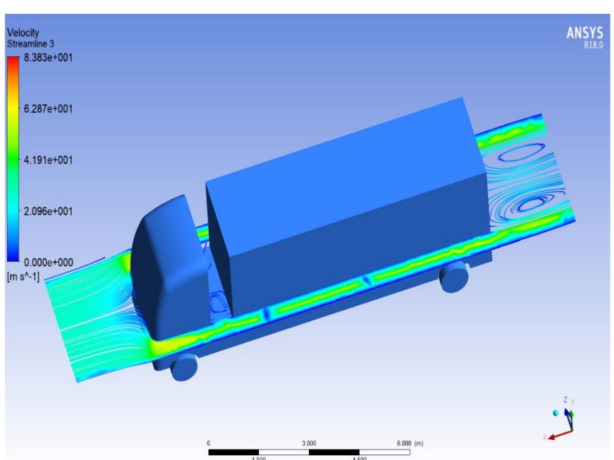


Fig – 14: Velocity Streamline (with side vanes)

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 VIII Aug 2021- Available at www.ijraset.com

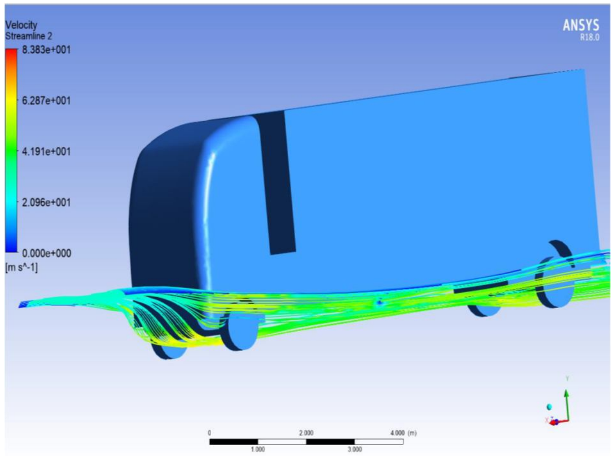


Fig – 15: Velocity Streamline (With front splitter)

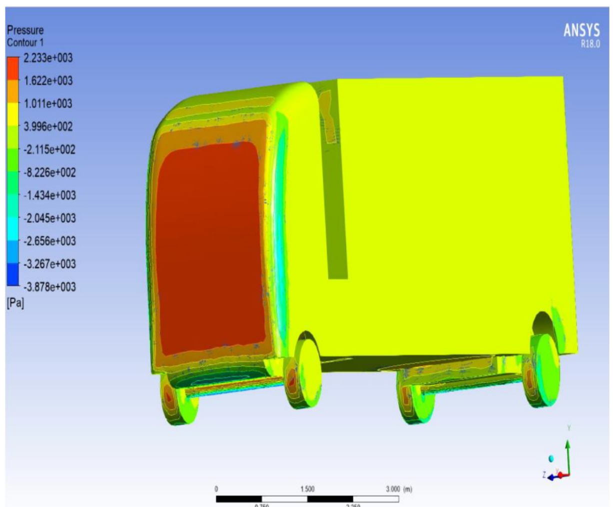


Fig – 16: Pressure Contour (with front splitter)

The total aerodynamic force obtained and the coefficient of drag on modified model is shown in the following figure.

Forces (N)	
Pressure	32506.445
Viscous	433.82642
Total	32940.272
Coefficients	
Pressure	7.2068
Viscous	0.0961
Total	7.3029

Table -2: Drag force and Coefficient (Modified Model)

Lastly, to postpone the separation of air by converging the air together to regulate the turbulence, the trapezoidal plates (base flaps) were added at the bottom of the container. From the literature survey, it is known to us that the effect of the base flaps depends on the Reynolds number. The length of the flap and angle is chosen such that it reduces the drag optimally.

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VIII Aug 2021- Available at www.ijraset.com

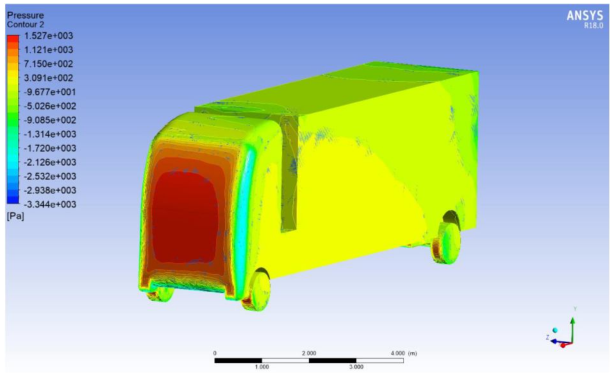


Fig – 17: Pressure contour (with base flaps)

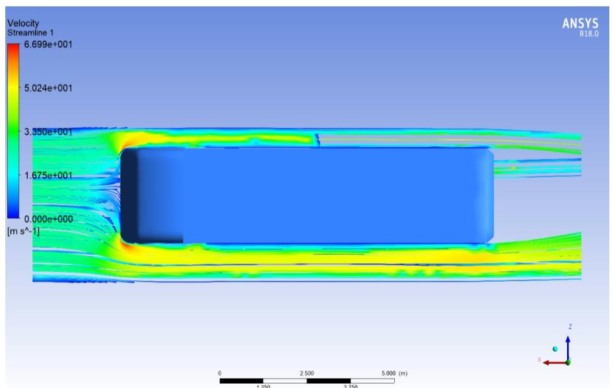


Fig – 18: Velocity streamline (Top View)

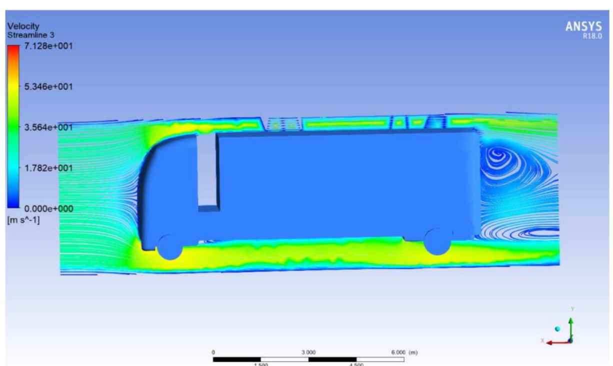


Fig – 19: Velocity streamline (Side View)

Forces (N)	
Pressure	9180.8008
Viscous	302.5520
Total	9483.3528
Coefficients	
Pressure	2.0723
Viscous	0.0682
Total	2.140

Table -3: Drag Force and coefficient (with base flaps)

The model after adding all the parts discussed above shows the following results. The drag force observed is significantly decreased which leads the vehicle to be more fuel efficient.

VI. EVALUATION PARAMETERS

Evaluation of drag coefficient often requires wind tunnel experiments and can be prohibitively expensive if not impossible for large objects or systems. Computational Fluid Dynamics (CFD) aerodynamic analysis offers an alternative approach and can be used as a very effective design tool in many industries. The boundary condition at the inlet is set at 27 m/s at 25° Celsius. The velocity and pressure contours are being developed with respect to these conditions.

VII. APPLICATION

Trailer aerodynamic devices help to increase fuel efficiency by lowering air resistance so that it takes less fuel to move down the road as speed increases. The per-vehicle fuel economy benefit of trailer aerodynamic devices can be high, ranging from 1% to over 10%, depending on the devices chosen.

Little gains in fuel economy improve profitability and help the environment. Two ways to make the truck more efficient are to use low rolling resistance tyres and to add aerodynamic aids. Besides choosing an efficient engine, one of the easiest ways to improve fuel-efficiency is to reduce drag by making the vehicle more aerodynamic. When you do a fuel economy side by side comparison with a truck that is less aerodynamic, the differences are too great to ignore.

VIII. EXPERIMENTAL RESULTS

Due to limiting circumstances, the experiments could not be done physically. Therefore, it was only possible to work on the design and analysis part of the model. For doing the experiments, a model can be made which would then be subjected to the wind tunnel setup. The inlet and the outlet could be defined, and coloured air would be used for the purpose of visible streamlines. Moreover, sensors and other electronic equipment could be used to study the object’s interaction with the wind. With these data we could qualify the data we have obtained from the computational fluid dynamics. Thus, it would help in modifying the object for improved efficiency.

IX. CONCLUSION

Addition of the deflector decreases the drag coefficient significantly, however the angle of the deflector should be carefully selected. Significant drag reduction is noticed by adding two cab vane corners at both frontal edges of cab. The base flaps help to reduce the turbulence at the bottom of the container by postponing the convergence of the streamlines. The front splitter at the frontal area helps to decrease the drag which would be observed due to underbody parts. After the installing all the parts, the aerodynamic drag was significantly reduced by approximately 45 to 50 percent. Since the actual vehicle would have a greater number of parts equipped the aerodynamic drag reduction will vary from vehicle to vehicle.

REFERENCES

[1] DOMINY R G, CORIN R J, HE L. A CFD investigation into the transient aerodynamic forces on overtaking road vehicle models [J]. Journal of Wind Engineering and Industrial Aerodynamics, 2008, 96: 1390−1411.

[2] Aerodynamic drag reduction of heavy vehicles using append devices by CFD analysis, Central South University Press and Springer-Verlag Berlin Heidelberg 2015

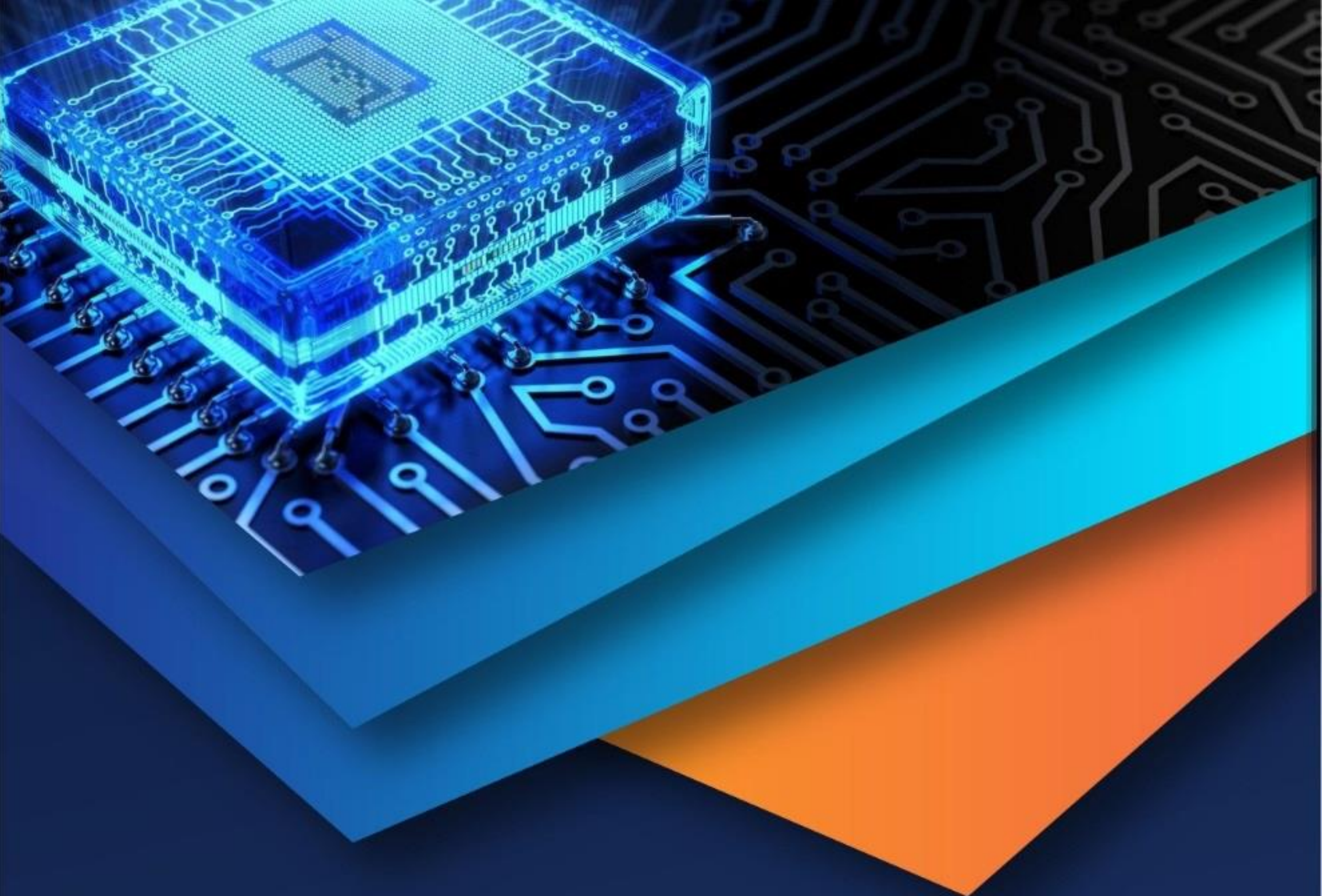
[3] ENGLAR R J. Advanced aerodynamic devices to improve the performance, economics, handling and safety of heavy vehicles [R]. SAE Technical Paper, No. 2001−01−2072

[4] ROCCHI D, ROSA L, SABBIONI E, SBROSI M, BELLOLI M. A numerical–experimental methodology for simulating the aerodynamic forces acting on a moving vehicle passing through the wake of a bridge tower under cross wind [J]. Journal of Wind Engineering and Industrial Aerodynamics, 2012, 104/105/106: 256−265.

[5] Aerodynamics of Road Vehicles: From Fluid Mechanics to Vehicle Engineering.

[6] Modifying the Aerodynamics of Your Road Car: Step-by-step Instructions to Improve the Aerodynamics of Road Cars JULIAN EDGAR.

[7] Aerodynamic Drag Mechanisms of Bluff Bodies and Road Vehicles.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY