RESEARCH ARTICLE OPEN ACCESS

AERODYNAMIC DRAG REDUCTION OF ERV's (Emergency Response Vehicle)

Meet Dubey[1], Samarth Parmar[2], Garv Patel[3], Vishal Tolani[4],
Asst. Prof. Priyanka Sharma*
[1] (Department of Automobile Engineering, Indus University, Ahmedabad
[2] (Department of Automobile Engineering, Indus University, Ahmedabad
[3] (Department of Automobile Engineering, Indus University, Ahmedabad
[4] (Department of Automobile Engineering, Indus University, Ahmedabad
*(Department of Automobile Engineering, Indus University, Ahmedabad

ABSTRACT
Engineers have been pushed to consider strategies for improving fuel economy as a result of recent increases in fuel prices and concerns about fuel emissions. Researchers are focused on the subject of vehicle's external aerodynamics to improve fuel efficiency, which involves minimizing drag. This study discusses research into the aerodynamics of emergency response vehicles, with the goal of lowering drag. The CFD analysis is carried out with SOLIDWORKS Flow Simulation. With some changes in external design, we can observe a good influence on overall fuel consumption as a result of these enhancements, and they provide a big opportunity for increasing fuel economy and reducing emissions from emergency response vehicles.
Keywords - Aerodynamics, Drag, Fuel Consumption, CFD, Emergency response vehicles, Fuel efficiency.

Date of Submission: 05-04-2022 Date of Acceptance: 20-04-2022

I. INTRODUCTION

When a vehicle is in motion, it is subjected to various forces such as drag force, lift force, rolling resistance force, and so on. Of all the forces, the drag force is the most visible. Drag force reduces top speed, reduces fuel economy, raises emissions, changes handling characteristics, and reduces acceleration. As a result, it is required to both reduce and boost efficiency. This can be accomplished by altering the operation of IC engines, using alternative and environmentally friendly fuels, or changing the outward design of automobiles using CFD (computational fluid dynamics). Since the adjustments to the IC engine have reached at such a point where additional modifications are no longer viable. It is also not a viable alternative due to insufficient development in eco-friendly fuel and the associated research costs. As a result, the most efficient and straightforward method is to alter the external design using CFD. The numerical simulation of the airflow around a car is usually complicated and difficult; therefore, use of CFD technique is an effective tool as it provides the detailed, quantitative data at any point in the flow field [1].External aerodynamics simulation is amongst most difficult and critical automobile CFD applications. CFD is becoming a useful technique in current fluid dynamics research, because of the rapid advancement of digital computers. It combines fluid mechanics, mathematics, and computer science fields [2]. CFD is used to tackle a wide range of research and engineering problems in a wide range of disciplines and industries, including aerodynamics and aerospace studies. Researchers from all over the world have been drawn to the activity of making car aerodynamics the best and, to be more precise, reducing drag, which is the most significant component of the entire project. The high usage of fuel has attracted several design engineers to enhance aerodynamics of vehicle. This study looks at how to reduce the aerodynamic drag of vans that have been modified to offer patient transport to hospitals as emergency response vehicles (ERVs). Aerodynamic enhancements for vans have received significantly less investigation [3].HGVs' aerodynamic drag is often decreased by streamlining airflow [4], minimizing the extent of wake and flow separation areas, and covering exposed under body parts, among other things. The most typical modification [5] is cab roof fairings, which try to optimize the flow from the cab to the trailer, but trailer front fairings [6] and boat tails and base flaps [7] have also been proved to be effective. Due to restrictions on characteristics such as maximum vehicle height and minimum vehicle height at the back in order to give adequate access to the vehicle, the design of ERVs cannot be modified drastically [8].The roundness of a bluff object's front corners and the degree of taper at its back end are two main elements that impact its drag co-efficient [9].

II. OBJECTIVE

- The Primary objective of this project is to improve fuel efficiency by performing CFD analysis on present Force van.
- Whereas emission reduction and increasing top speed are the secondary objectives of our project.
- By making some changes in external design of a van and reducing drag we can able to achieve our objectives.

III. METHODOLOGY AND VARIOUS STEPS

Figures 1 and 2 depict the vehicle's geometry, which is based on a simplified model of a Force van operated as an ambulance. The steps of CFD analysis are discussed below in detail.

3.1) MODELLING

This baseline model is one of the models of Force van. The body of van was designed to actual van dimensions which are almost common to the actual design.

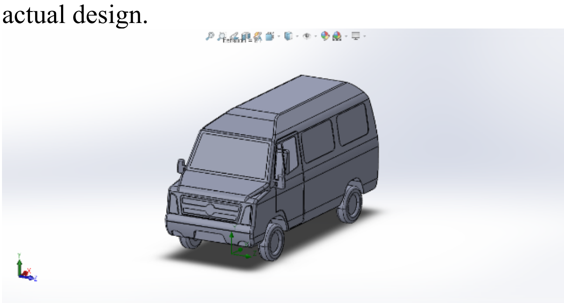

Fig. 1

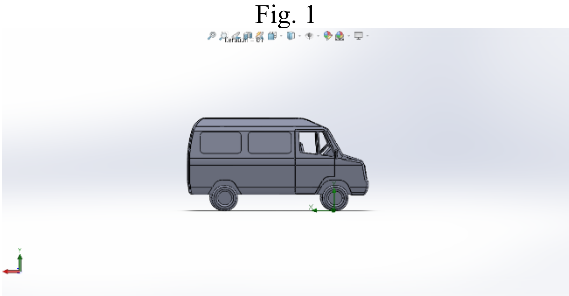

Fig. 2

3.2) MODEFIED MODEL

This model has modified at its outer design. This model is designed under the consideration of aerodynamic aspects and parameters.

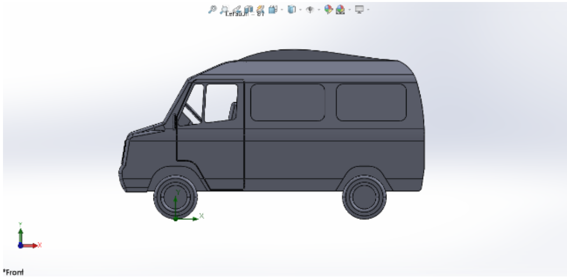

Fig. 3

3.3) MESHING

Meshing is the process in which the continuous geometric shape of an object is broken down into thousands or more of shapes to properly define the physical shape of the object [10]. The more intricate the mesh, the more precise the 3D CAD model, enabling high-fidelity simulations. SOLIDWORKS Mesh Tool was used to create a suitable mesh. As illustrated in Fig.4, the mesh was coarser in the inner domain and finer in the contact zone with the vehicle.

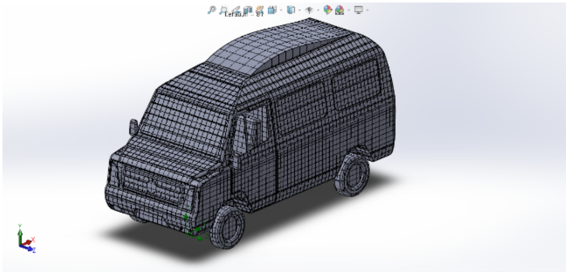

Fig. 4

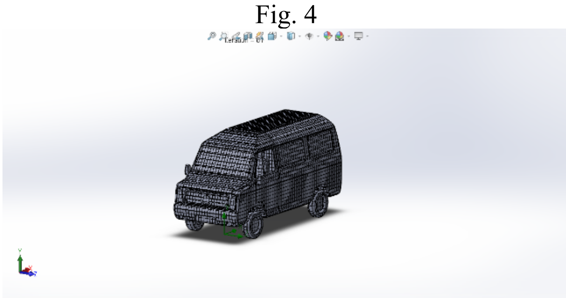

Fig. 5

3.4) BOUNDARY CONDITIONS

On the mesh model, boundary conditions were imposed. Only straight wind conditions were considered in the simulation at vehicle speed 80 km/hr. The constant velocity intake condition was used at the inlet to simulate constant wind velocity circumstances. At the outlet, zero-gauge pressure was applied, and the working pressure was set to atmospheric pressure.

IV. RESULT AND DISCUSSION

In this section, we have discussed the analysis results.

4.1) STATIC PRESSSURE PLOT

Baseline model

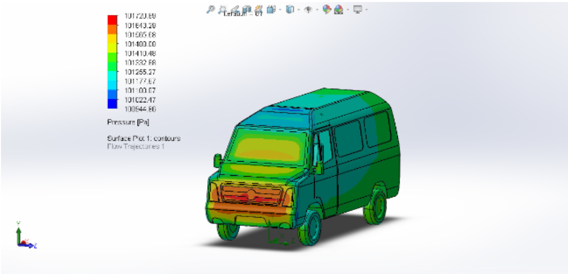

Fig. 6

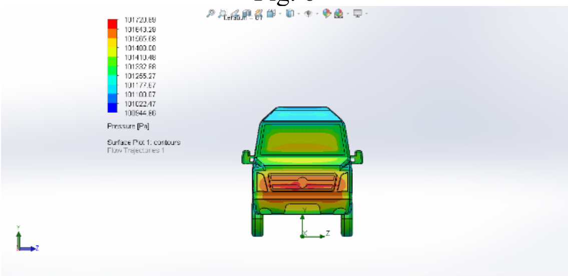

Fig. 7

Modified model

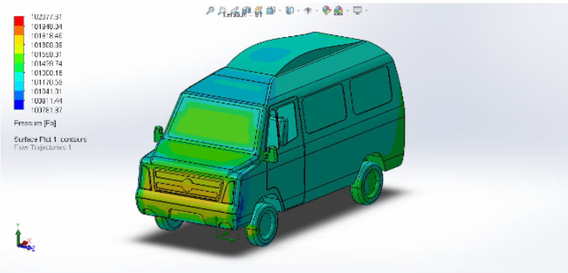

Fig. 8

From both figures it is evident that the stagnation point in the front as air flow strikes at frond end and brought to the rest.

4.2) DRAG CO-EFFICIENT

For our Model the force in opposite direction of motion i.e., Drag force is in x-axis.
The Formula for Drag Coefficient is
Cd= (2* drag force)/ ρAV^2
we get its value as 0.31 for baseline model and similarly for modified model we get value 0.26.

4.3) VELOCITY PLOT

Figure 9 and 10 show the contours of the velocity for the original and modified design respectively. The blue colour in the graphics denotes a low-pressure region, which is the primary source of drag. This region is just at the rear of the baseline car and is caused by the separation and circulation of the flow.

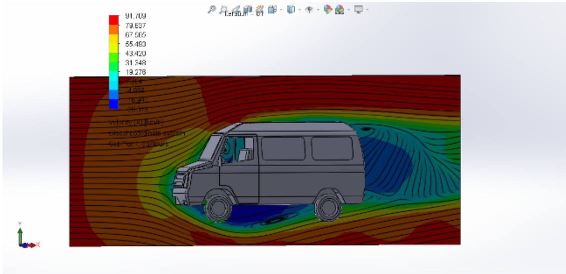

Fig.9

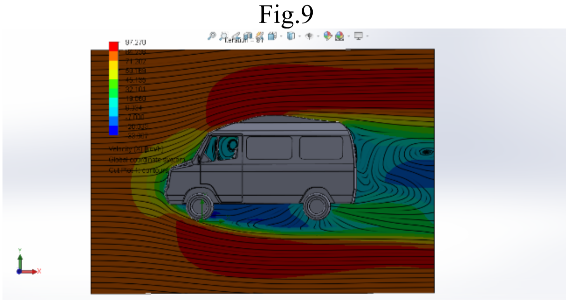

Fig. 10

In the modified case, the wake region (blue colour) vanishes, and the size of the low-pressure separation area at the rear of the vehicle is reduced since the flow is more directed towards the ground due to the revised roof design. As a result, the drag value decreases.

V. CONCLUSION

Rising fuel prices, along with the desire to decrease the environmental impact of fleet operations, are stimulating interest in enhancing ERV aerodynamic design. The experimental and computational results indicate that enhancing the aerodynamic shape of the rooftop and light bars in ERV transformations offers a considerable possibility for minimizing ERV fuel usage [7].The velocity contour diagram clearly shows that changing the roof design reduces the pressure in front of the van and also eliminates the area of separation and low pressure, which are primarily responsible for excess drag production. So, we can say that by making some changes in external design we have reduced the drag coefficient by 16% and achieved our objectives.

REFERENCES

[1]. Mohan Jagadeesh Kumar M, Anoop Dubey, Shashank Chheniya, Amar Jadhav. *Effect of Vortex generators on Aerodynamics of a Car: CFD Analysis*, International Journal of Innovation in Engineering and Technology (IJIET).

[2]. *A Numerical Study on Rear-spoiler Of Passenger Vehicle*. Xu-xia Hu, Eric T.T. Wong, World Academy of Science, Engineering and Technology 57 2011.

[3]. Gilkeson CA, Toropov VV, Thompson HM, Wilson MCT, Foxley NA, et al. (2013) *multi-objective aerodynamic shape optimization of small livestock trailers* Engineering Optimization 45: 1309-1330.
[4]. http://www.donbur.co.uk/.
[5]. Leuschen J, Cooper KR (2009) *Summary of full-scale wind tunnel tests of aerodynamic drag-reducing devices for tractor-trailers*. The Aerodynamics of Heavy Vehicles II: Trucks, Buses and Trains. Springer. New York.
[6]. Watkins S, Saunders JW, Hoffman PH (1993) *Comparison of road and windtunnel drag reductions for commercial vehicles*. Journal of Wind Engineering and Industrial Aerodynamics 49: 411-420.
[7]. Cooper KR (2003) *Truck Aerodynamics Rebornlessons from the past*. SAE Paper 9-19.
[8]. Taherkhani, deBoer, Gaskell, Gilkeson, Hewson, Keech, Thompson and Toropov. *Aerodynamic Drag Reduction of Emergency Response Vehicles*. Advances in Automobile Engineering.
[9]. Akshoy Ranjan Paul, Anuj Jain and Firoz Alam. *Drag reduction of a passenger car using flow control techniques*, International Journal of Automotive Technology, Vol. 20, No. 2, pp. 397−410 (2019) DOI 10.1007/s12239−019−0039−2
[10]. https://simutechgroup.com/why-is-meshingimportant-for-fea-fluid-simulations/

Meet Dubey, et. al. "AERODYNAMIC DRAG REDUCTION OF ERV's (Emergency Response Vehicle)." *International Journal of Engineering Research and Applications (IJERA)*, vol.12 (04), 2022, pp 36-39.