JOURNAL ON COMMUNICATIONS

ISSN:1000-436X

REGISTERED

Scopus

www.jocs.review

CFD ANALYSIS OF A HEAVY TRUCK FOR REDUCTION OF DRAG AND FUEL CONSUMPTION BY VARYING VEHICLE VELOCITIES

M. Sri Rama Murthy[1], K.Rambabu[2], M. Hareesh[3], K. Tejeswara Rao[4]

[1] *Professor, Department of Mechanical Engineering, Sir C.R.Reddy College of Engineering, India*
[2] *Professor, Department of Mechanical Engineering, Sir C.R.Reddy College of Engineering, India*
[3] *Student, Department of Mechanical Engineering, Sir C.R.Reddy College of Engineering, India*
[4] *Student, Department of Mechanical Engineering, Sir C.R.Reddy College of Engineering, India*

ABSTRACT

In order to determine the relationship between vehicle speeds and fuel reduction related to truck aerodynamic technologies, heavy vehicle trucks were used for testing. In this study, the CFD analysis of the heavy truck has been performed using ANSYS software for reduction of drag and fuel consumption. Continuous speed testing was done at various speeds on the road and results are verified. Three different vehicle speeds 35, 55, and 75 KMPH were used to gather data. To determine the fuel consumption at different vehicle speeds, two designs are considered for analysis that is existing and proposed designs by adding extra features like Deflector, Vanes between Cab & Container and Base Flaps at End of Truck. By using these designs heavy vehicle truck was modelled using CATIA software and computational fluid dynamic analysis was done using ANSYS software. The main intention behind this project is to compute the Drag coefficient and fuel consumption for three different vehicle speeds and to compare the results. Further, the study involves finding out the best efficient design that gives low drag coefficient.

Keywords: *Heavy vehicle truck, fuel consumption, vehicle speeds, aerodynamics, CFD.*

1. INTRODUCTION

Nowadays, demand for reducing the fuel consumption of vehicles within the automotive industry is increasing. Heavy commercial vehicles in comparison to other ground vehicles, due to high aerodynamic drag, have low fuel efficiency. Recent researches about fuel reduction technologies for trucks show that aerodynamic improvement is one of the most important technologies when it comes to fuel saving. Any reduction of aerodynamic drag will result in significant fuel savings and reductions in greenhouse gas emissions. 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. 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.

From the literature, a CFD study was conducted to design and optimize the cabin geometry and its various parts for drag reduction including the side deflectors, the mirrors and the sun visor. For the validation of computational results, an experimental investigation using a 1/5 scale truck model was conducted in a wind tunnel. Both steady and unsteady CFD simulations were performed. The comparison of steady/unsteady results revealed that the time-averaged unsteady flow characteristics were practically the same as the steady calculations for design purposes. Also, the effectiveness of adding a fairwater on the top of truck's head was investigated for different shrink angles. In this work, a comprehensive CFD study was conducted to investigate the influences of supplementary parts on drag reduction of heavy commercial vehicles.

[1] **L. Ananthan Raman et.al. (2016)** had conducted a study on aerodynamic drag reduction to reduce fuel consumption in vehicles. They conducted passive tests on a SUV model by extending its rear end (rear fairing), adding a rear plate (rear screen) and by adding a vortex generator (Delta wing and bump shaped). A 6.5% and 26% reduction of drag was found by installation of rear screens and rear fairing respectively. Among the vortex generators, the delta wing type was found more effective for drag reduction.

[2] A work on reduction of Aerodynamic Drag in generic trucks using geometrically optimized rear cabin bumps was carried out by **Abdullah Ait Moussa et.al, (2015).** They used a 1/10th scaled half model of a generic truck and added three equally spaced bumps on the top of the cabin surface. Thereafter they used Taguchi or Orthogonal array optimization method to study the effect of these bumps on drag.

[3] **SHOBIT SENGAR et.al, (2014)** conducted wind tunnel experiments to determine forces acting on three types of vehicles i.e. Ambassador, Aventador LP 700-4 and F1 car .and compared for the best aerodynamic features. These models are tested under different wind conditions. It was found that the F1 car is the most aerodynamic amongst the three followed by the Aventador and then Ambassador. Also, the linings of the coupe help in channelizing the air when the vehicle is in motion which leads the air to the rear end where spoilers are provided which provides additional stability at high speeds.

[4] Wind tunnel and ANSYS fluent tests were conducted by **Abdul Kareem SH. Mahdi – Obeidi et.al, (2014)** on an open wheel race car made by students of Taylor's University. They studied the effect of Radiator air channel in drag optimization and compared both experimental and numerical results. From the results, it was found that there is a reduction

of drag to 0.563 from 0.619, when the angle of tilt of radiator channel is increased from 36˚ to 72.5˚. There was only 7.7% between both results.

[5] **J. Abhinesh et.al., (2014)** conducted a CFD analysis of two Volvo intercity truck models i.e. existing and modified one, to reduce drag and fuel consumption. After the study it was found that the drag reduction is of about 10%. The Original model's coefficient of drag was found to be 0.8 and for the modified model was found to be 0.7.

[6] Low speed wind tunnel testing was performed by **Ashfaque et.al, (2014)** to discuss the Drag force analysis of a car. Here velocities of fluid, drag & lift forces were determined. It was observed that the design of low-speed wind tunnel is somewhat different to other wind tunnel. The machine was useful for educational and research purpose.

[7] FRANCESCO MARIANI et.al, (2012) have conducted numerical analysis on a race car model which was developed by the students of University of Perugia. Their experiment was focused on changing the design of car nose so as to optimize the aerodynamic drag. They called the original model as "A" and modified as "B". In model B they added a front wing, modified the head rest, adopted an air extractor and added a wing on front tire.

[8] Wind tunnel testing was performed by **UPENDRA S. ROHATGI et.al, (2012)** on the model of General Motor SUV for different wind conditions and road clearance. In this work, the researchers were considered two passive devices, one is rear screen plate fixed behind the car and the other one is rear fairing where the end of the car is aerodynamically extended. The conclusion was that rear screen could reduce drag up to 6.5% and rear fairing can reduce the drag by 26%. It was also concluded that efficiency of rear screen equally depends on configuration, dimensions and arrangement of screens.

[9] YINGCHAO ZHANG et.al, (2009) have narrated the details of the virtual wind tunnel test simulation in this paper. Drag coefficient, velocity contour and pressure distribution were determined using this test. It was found that it is a simple, effective, convenient and fast way to do aerodynamic numerical simulation based CFD in the process of car styling.

[10] Investigation has been performed by **KEISUKE NISUGI et.al, (2004)** to reduce the Aerodynamic Drag of a vehicle with the help of feedback flow control mechanism. In their study, sensor and controller system was mounted on the vehicle to sense the information about velocity and pressure components. As per the requirement the controller drives the actuator which in turn operates the control port where blowing and suction of air takes place. With this, there is a reduction of 20% drag as compared to the vehicle without feedback flow control system.

[11]ROSE MCCALLAN et.al., (1999) have conducted Wind tunnel testing on the models of 1:14 Class 7 and Class 8 heavy duty Sandia trucks to reduce the drag and fuel consumption.. The PIV (Particle image velocimetry) measurements were taken in the model wake. Oil film interferometer techniques (OFI) for measuring skin friction and pressure sensitive paint (PSP) measurements were also used. They found that PIV approach to calculate various parameters in the Wind tunnel can be effective in finding more precise and accurate results

[12] Investigations on drag reduction for four models of car have been carried out by **R.H. HEALD. et.al, (1933)** and comparisons were made with the model which was 10 years old. From this work, it was known that elimination of fenders and other projections together with pronounced fairing of body of one model reduced the drag coefficient quite significantly. Further, an additional decrease in value of drag coefficient was observed by eliminating the windshield and fairing the whole body of car.

2. METHODS AND MATERIALS

2.1 PROBLEM DEFINITION

The main objective of drag reduction is to obtain better fuel consumption of the truck and increase in the operational performances. This project is about the study of modelling and flow analysis of heavy truck with two different modified truck designs, concern with calculations of fuel consumption and drag reduction. The truck designs were developed in CATIAV5 R20 with in standard geometry. And the analysis was carried out for the designed model at speeds 35kmph, 55kmph, 75kmph by using CFD.

2.2 METHODOLOGY

In this study numerical simulations of different vehicle configurations are performed. Starting with evaluating the results of the baseline simulation, configurations of interest can be identified. For each configuration the CFD process described below is performed. The CFD process can be divided into three steps; pre-processing, solving and post-processing.

Here simulations are conducted by evaluating different drag reducing trailer devices by means of CFD. ANSYS and CATIA are the software used and the setup and essential settings are described. Further on, from the evaluation of the individual devices, combinations of these are tested and analysed to see if drag can be improved even more.

2.3 OBJECTIVE

The objective of this work is to conduct a thorough analysis of current Omni bus aerodynamics through computer modelling and provide recommendations for improvement.

- To model an existing Omni bus, as a baseline model using CATIA modelling software.
- To perform the flow analysis on the baseline model using CFD tool fluent.
- To design new model of bus such that drag force is reduced.
- To perform flow analysis on the new model.
- To achieve better fuel efficiency by reducing the drag force.

Deflectors, vanes and base flaps are used for reduction of Drag and improving the efficiency.

Fig. 2.1: Deflector

Deflectors, commonly known as wind deflectors, on heavy vehicles such as trucks, buses, or construction equipment, offer several advantages: Deflectors are a cost-effective and practical modification that can enhance performance, safety, and comfort for heavy vehicles.

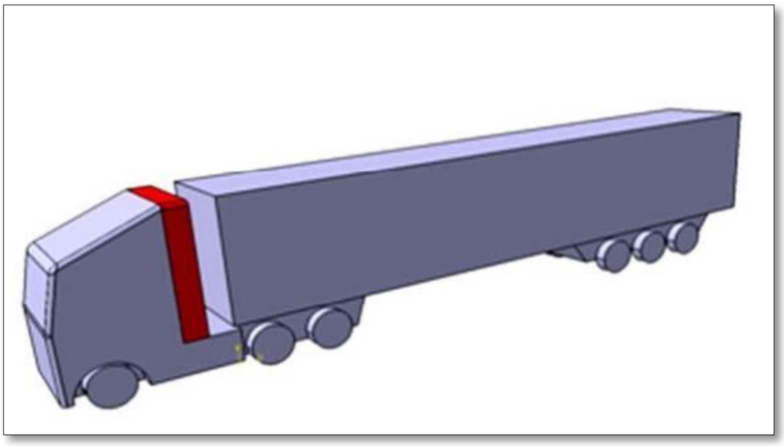


Fig.2.2: Vanes at gap between cab & container

Vanes at the gap between the cab and container optimize the airflow around the vehicle, improving fuel efficiency, safety, and stability while also prolonging the vehicle's life by reducing wear caused by turbulence and drag.

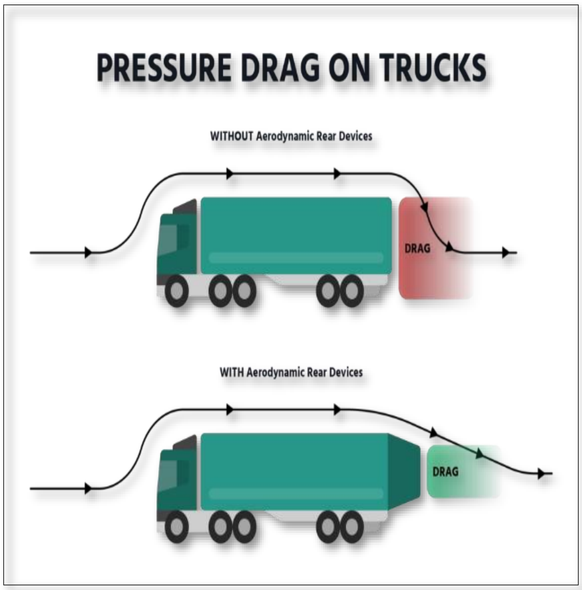
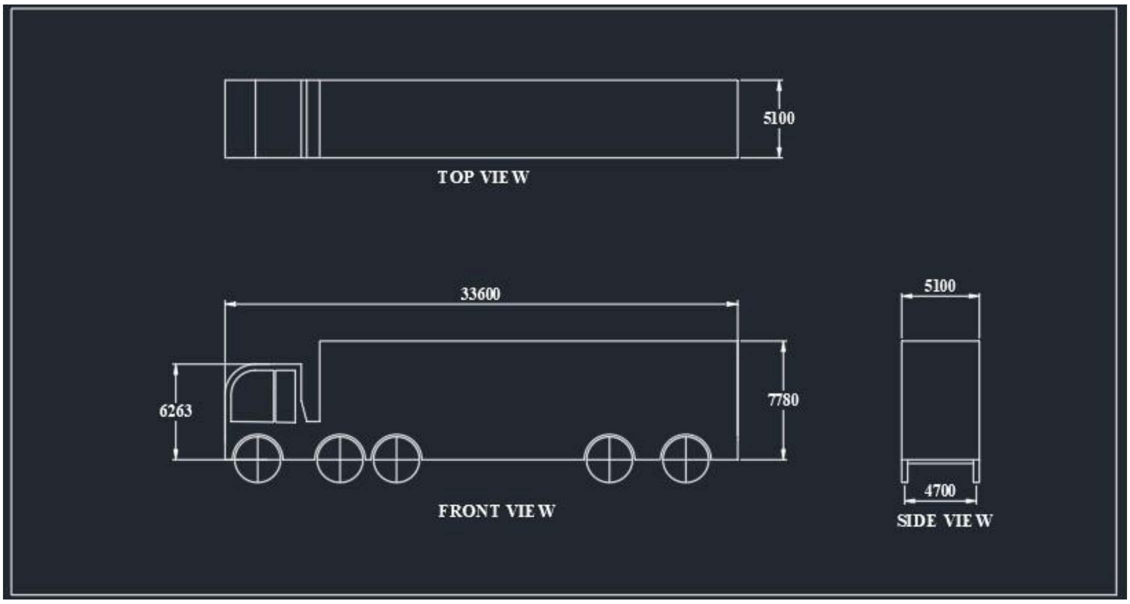


Fig. 2.3: Base flaps

Base flaps on heavy vehicle trucks offer significant aerodynamic advantages, primarily by reducing drag and improving fuel efficiency, while also protecting vehicles behind them from debris.

2.4 DESIGN PROCEDURE USING CATIA V5

- **EXISTING MODEL**



Fig, 2.4: 2D Drawing of Existing Truck

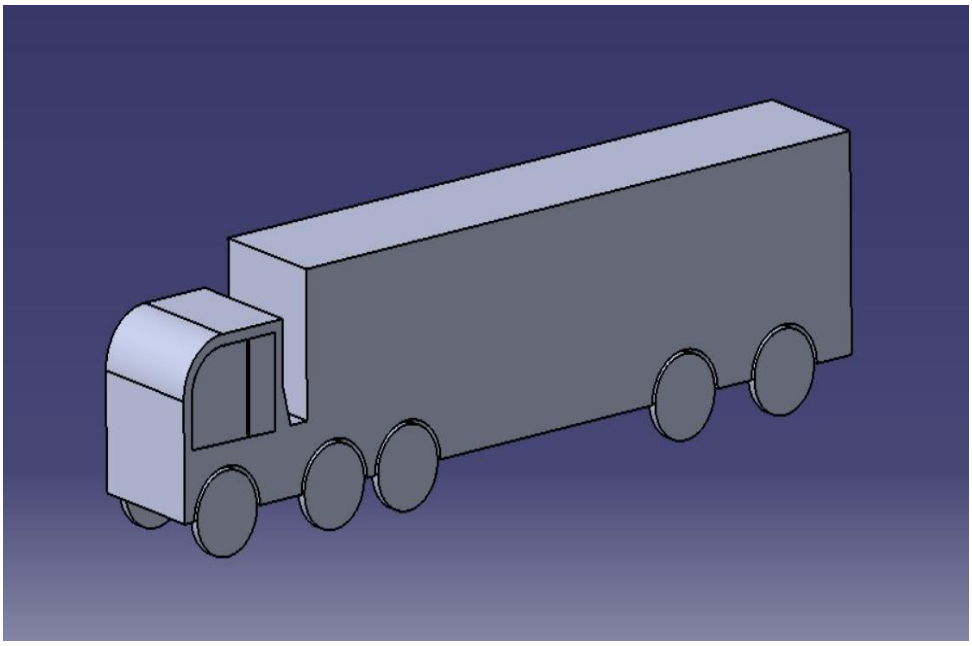


Fig.2.5: 3D Design of the Existing Truck

PROPOSED MODEL:

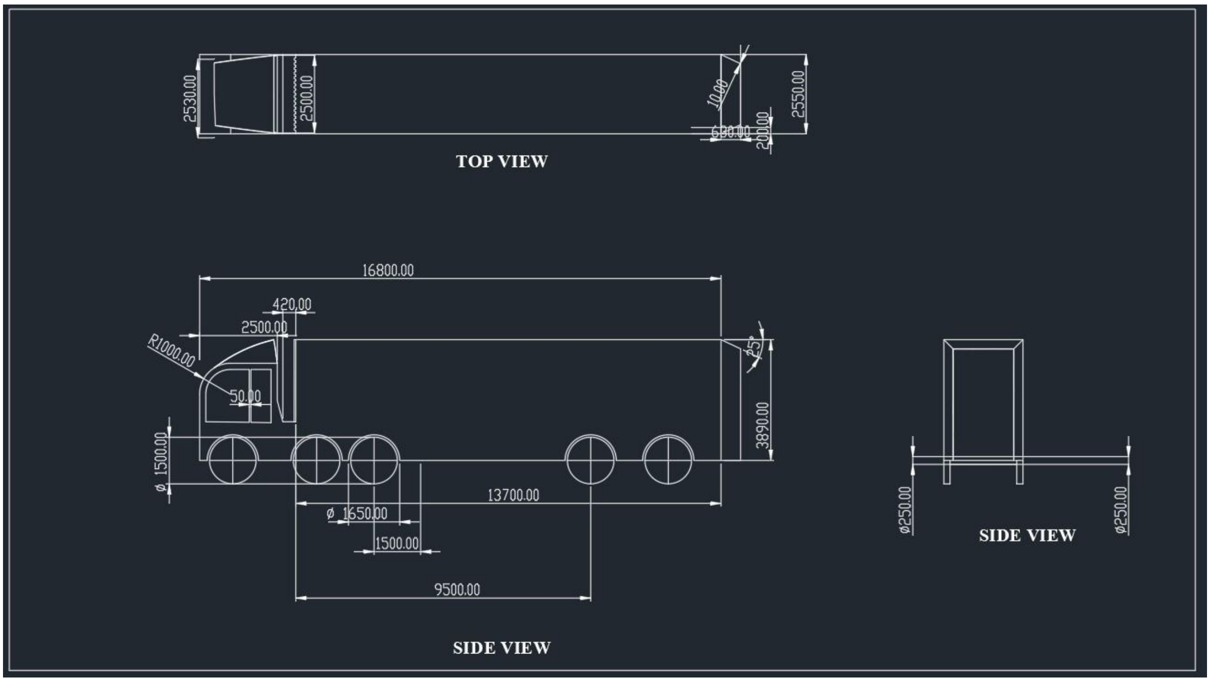


Fig.2.6: 2D drawing of Proposed Model of Truck

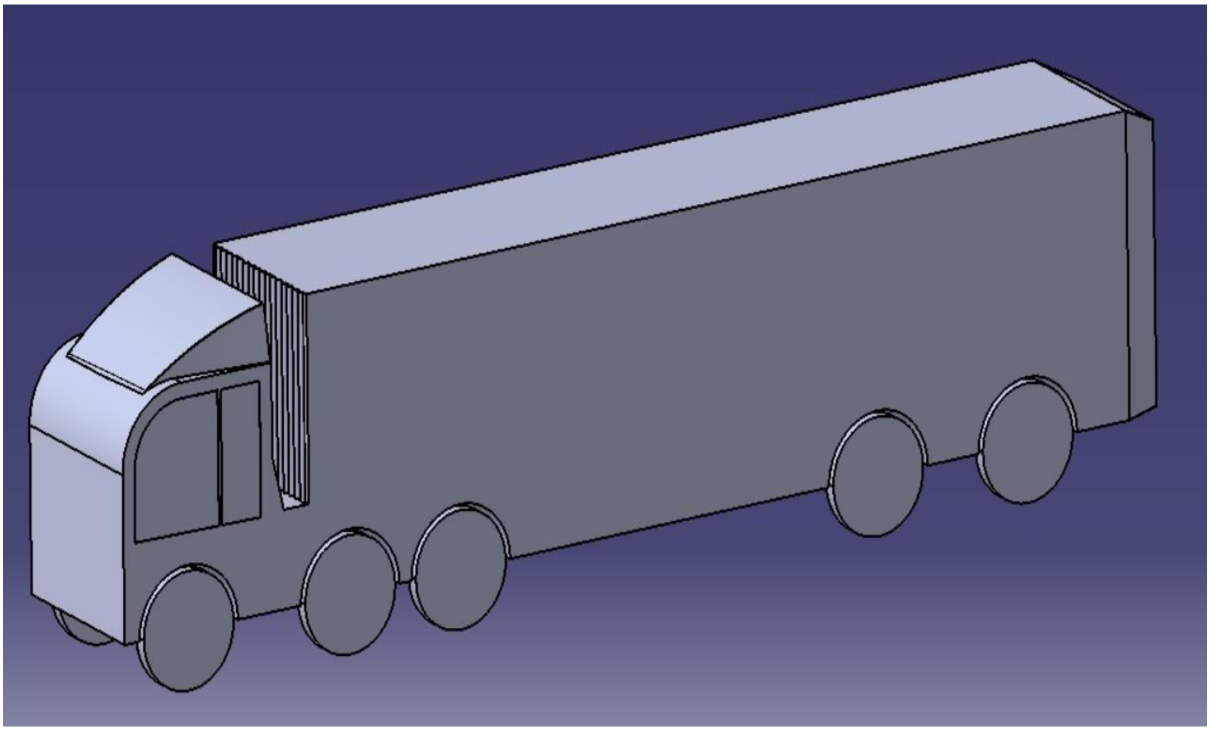


Fig.2.7: 3D Catia Design of Proposed Model of Truck

ANSYS is a large-scale multipurpose finite element program developed and maintained by ANSYS Inc. to analyse a wide spectrum of problems encountered in engineering mechanics.

MESHING:

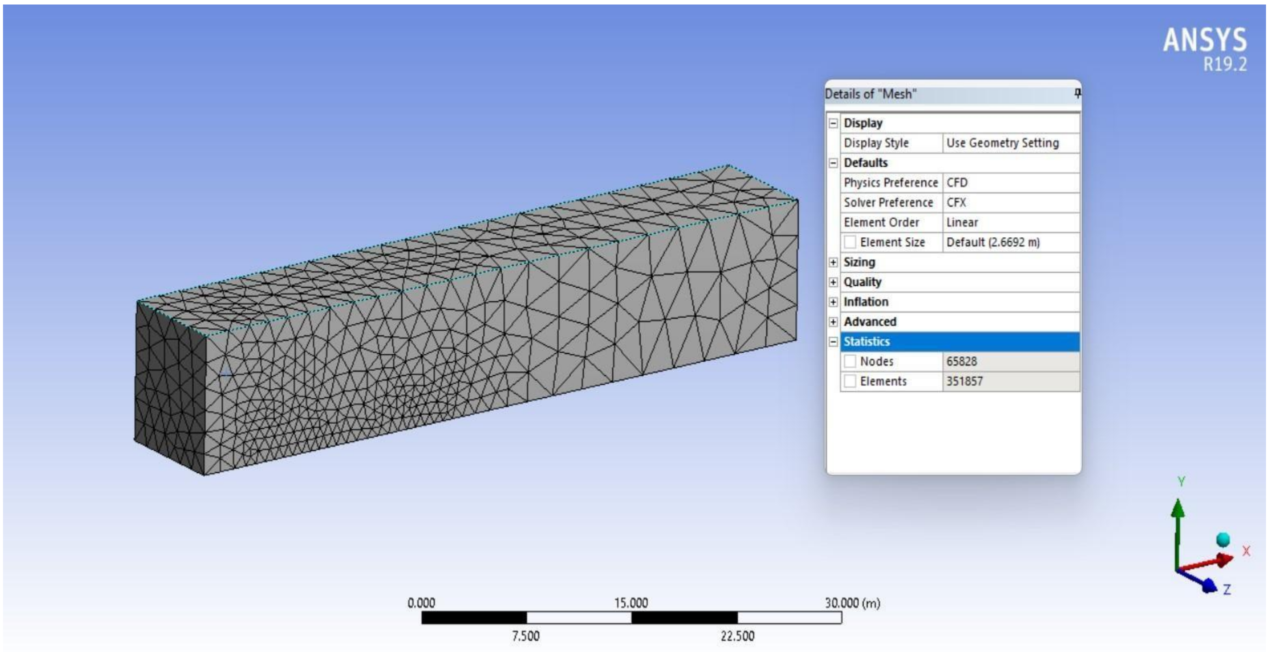


Fig.2.8: Meshing

BOUNDRY CONDITIONS:

Boundary conditions were applied on the meshed model. In the simulation only straight wind condition was considered at different vehicle speeds up to 75Km/hr. Constant velocity inlet condition was applied at the inlet to replicate the constant wind velocity conditions same as wind tunnel tests. Zero-gauge pressure was applied at the outlet with operating pressure as atmospheric pressure.

3. RESULTS & DISCUSSIONS

3.1PRESSURE VARIATIONS ON TRUCK

Following are the results obtained by the CFD Fluent simulation analysis conducted on the heavy vehicle truck at an optimal speed of 35 KMPH. The results of pressure acting on the truck were plotted below for both basic as well as advanced (proposed) model.

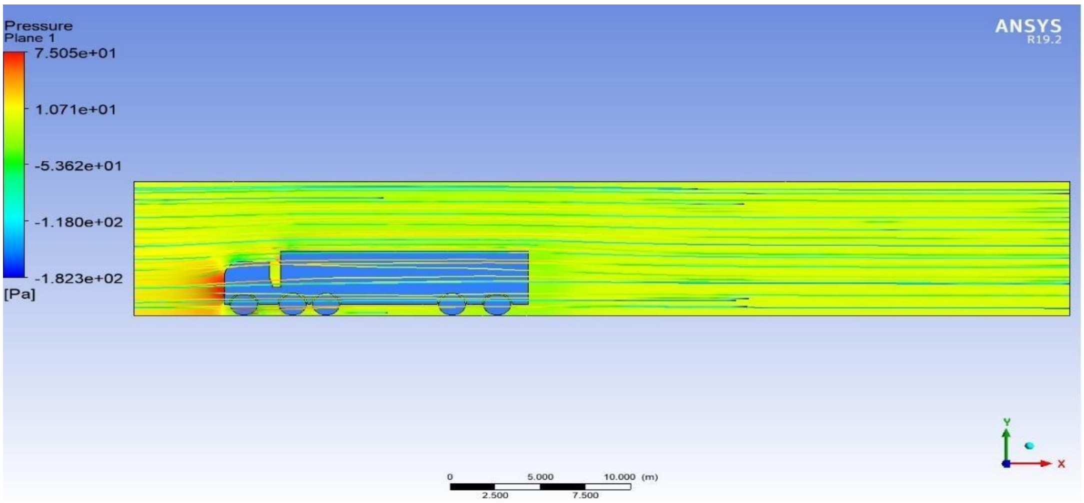


Fig.3.1: Pressure Acting on the Truck Basic Model @ 35 Kmph

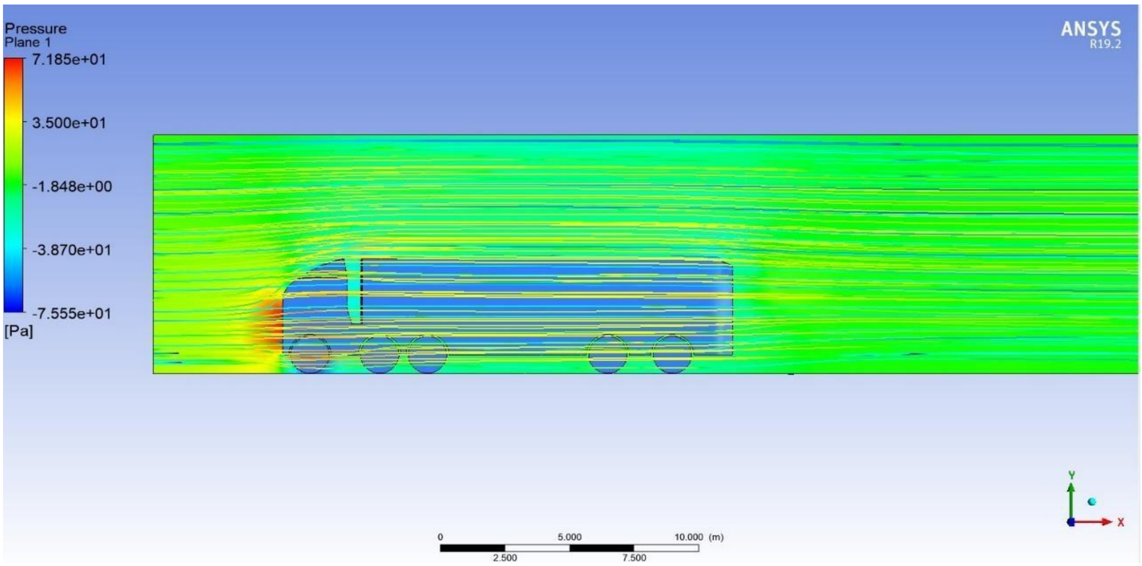


Fig.3.2: Pressure Acting on the Proposed Model @ 35 Kmph

3.2 VELOCITY VARIATIONS ON TRUCK

These are the results obtained by the CFD Fluent simulation analysis conducted on the heavy vehicle truck at an optimal speed of 35 KMPH. The results of velocity at U direction acting on the truck were plotted below for both basic as well as advanced (proposed) model.

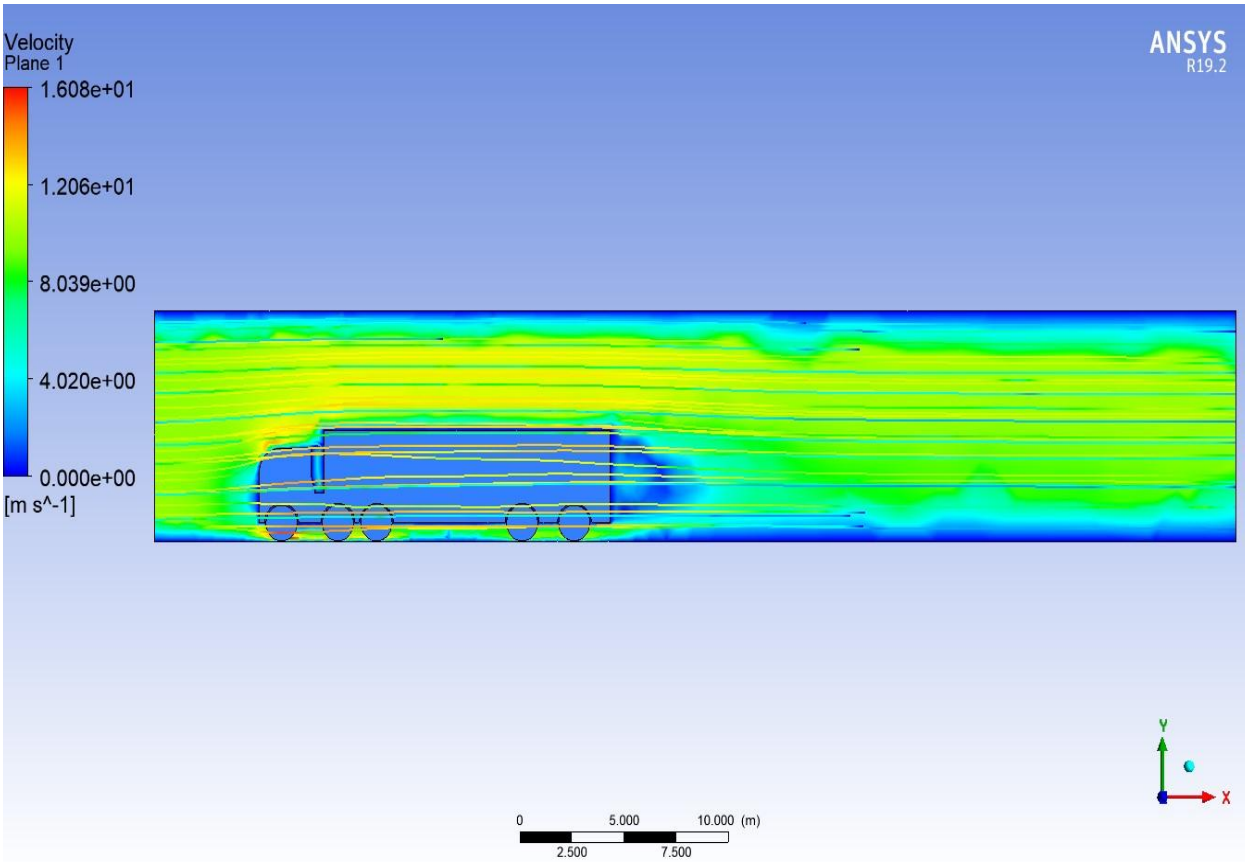


Fig.3.3: Velocity Acting on the Basic Model @ 35 Kmph

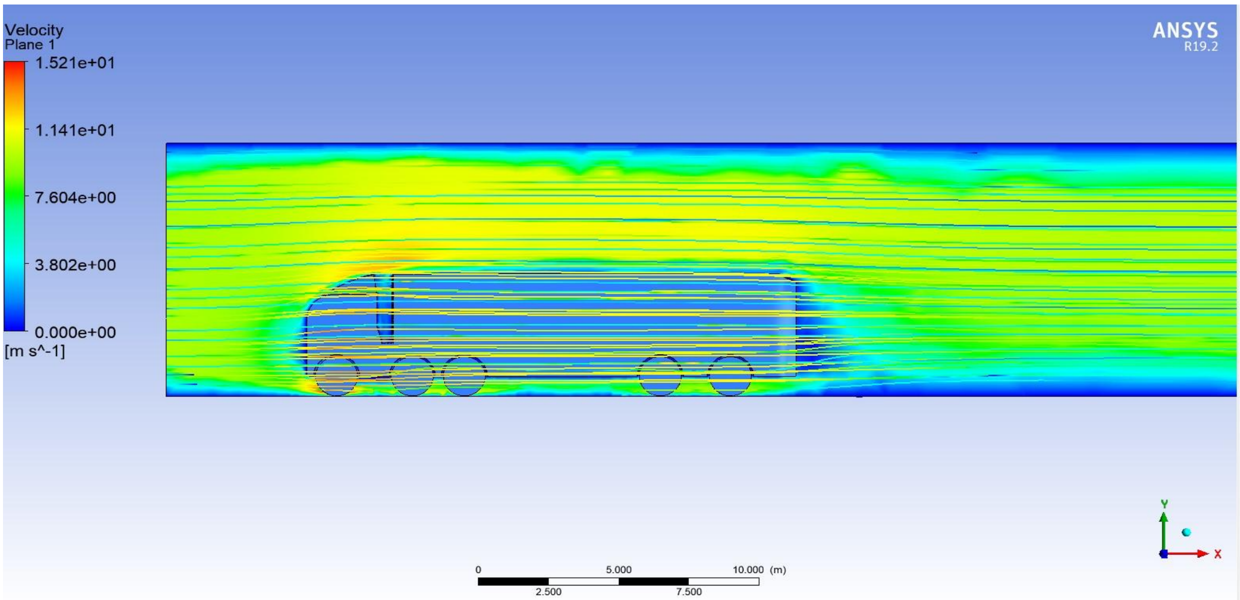


Fig.3.4: Velocity Acting on the Proposed Model @ 35 Kmph

3.3 VELOCITY AT U DIRECTION ON TRUCK:

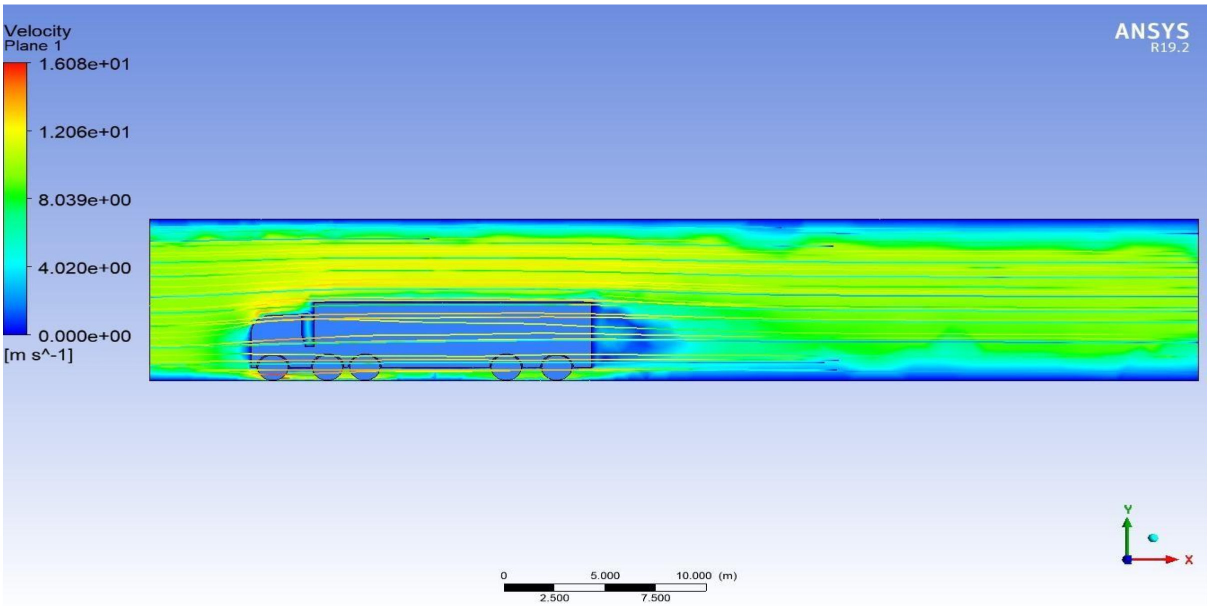


Fig.3.5: Velocity at U-Direction of Basic Model@ 35 Kmph

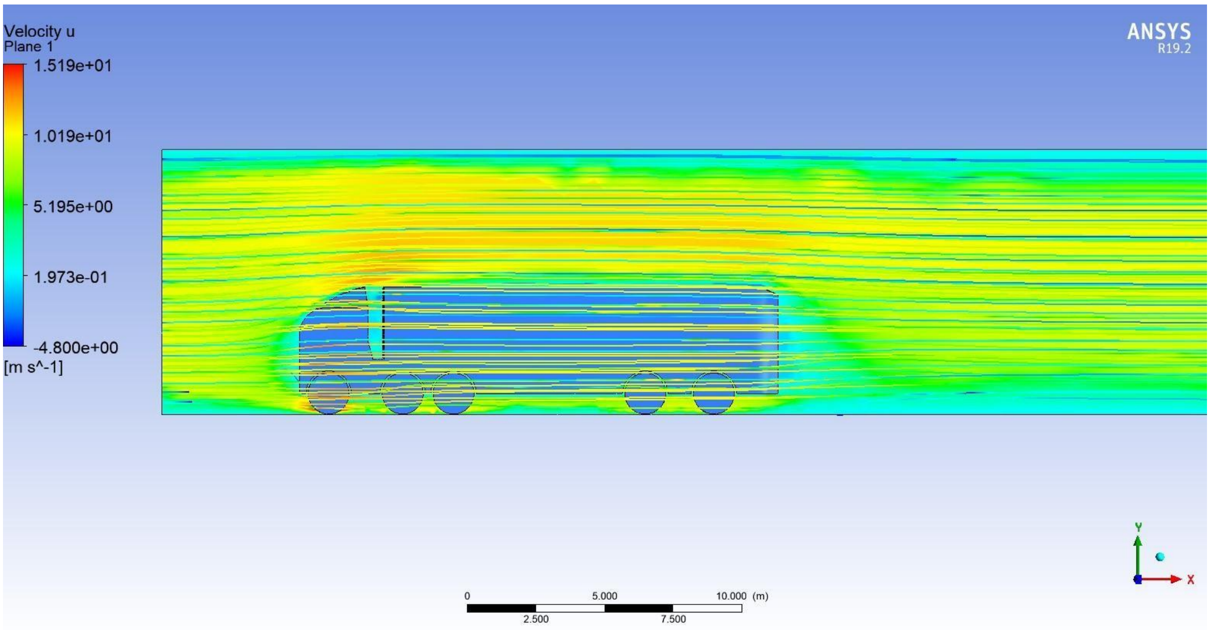


Fig.3.6: Velocity at U-Direction of Proposed Model@ 35 Kmph

3.4 VELOCITY AT V DIRECTION ON TRUCK

These are the results obtained by the CFD Fluent simulation analysis conducted on the heavy vehicle truck at an optimal speed of 35 KMPH. The results of velocity at V direction acting on the truck were plotted below for both basic as well as advanced (proposed) model.

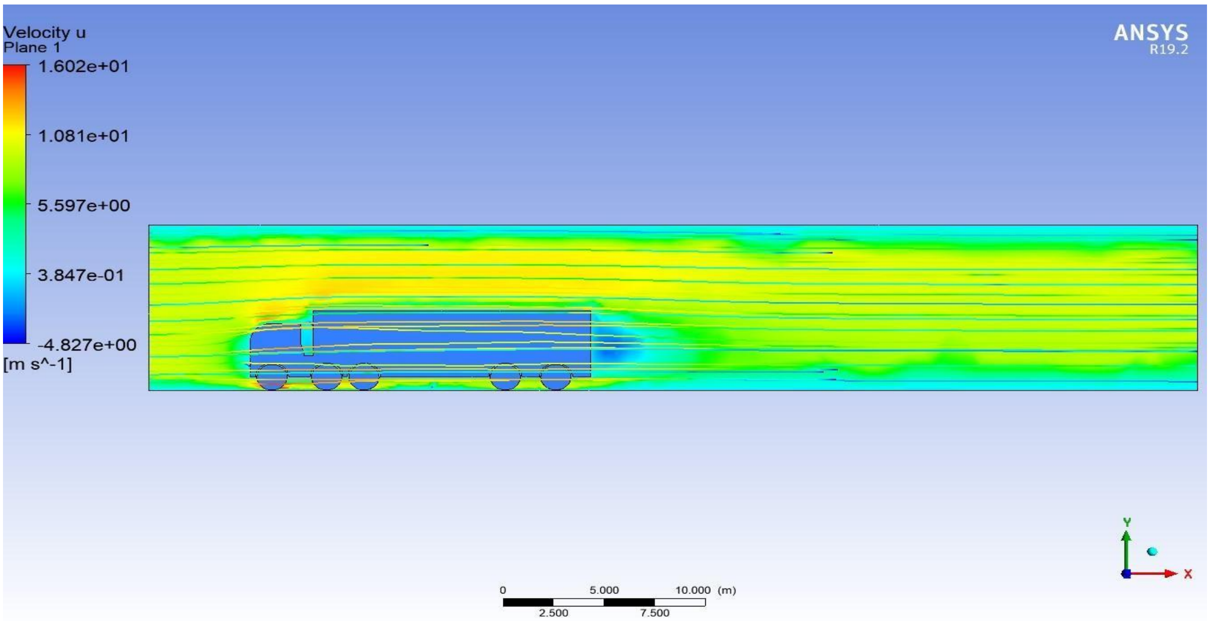


Fig.3.7: Velocity at V-Direction of Basic Model@ 35 Kmph

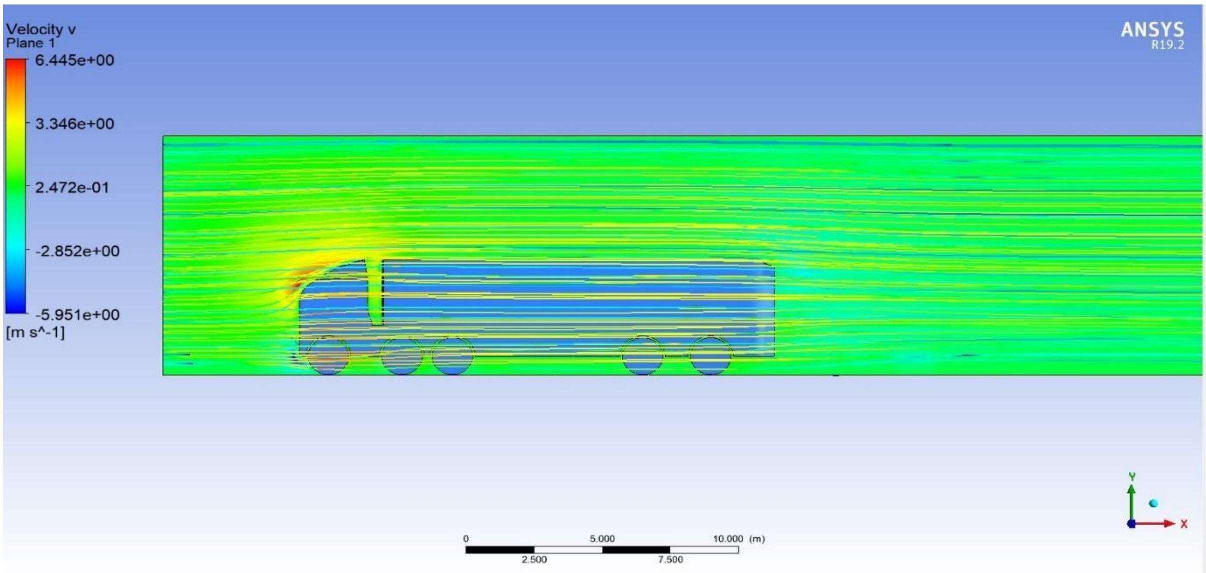


Fig.3.8: Velocity at V-Direction of Proposed Model@ 35 Kmph

Same analysis has been done at other speeds of 55 kmph and 75 kmph for both the models of truck i.e. existing and proposed. It has been observed that almost similar behaviour with respect to pressure and velocity variations as that of 35 kmph speed.

3.5 PRESSURE VARIATIONS AT DIFFERENT SPEEDS

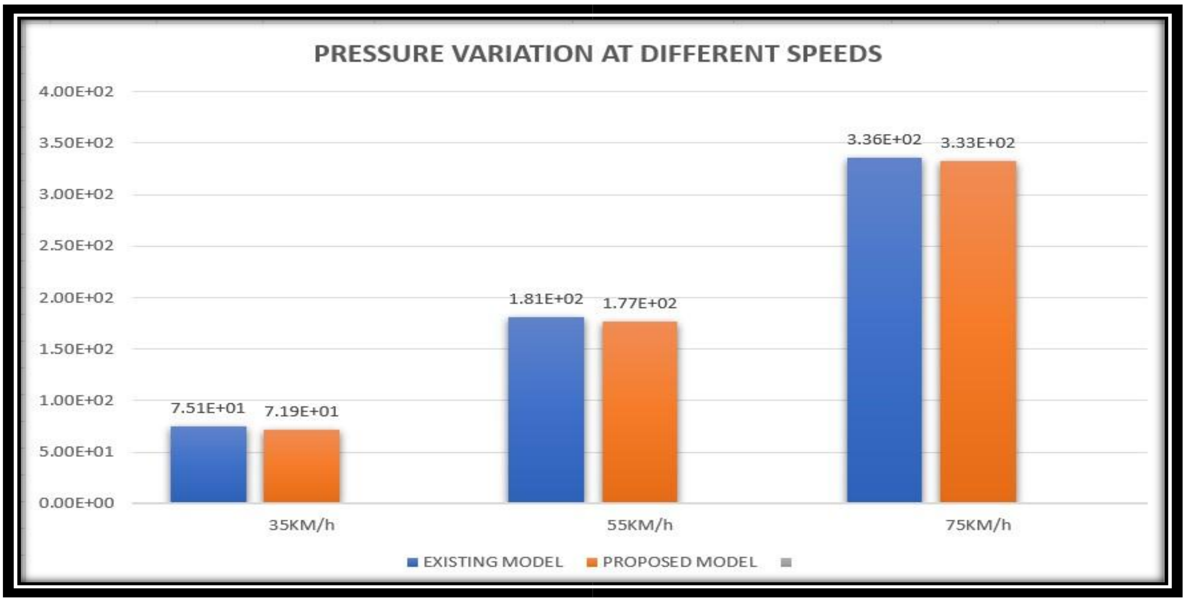


Fig.3.9: Pressure Variations at Different Speeds

Fig.3.9 shows the pressure variations for two models of trucks i.e., existing and proposed at speeds of 35kmph/55kmph/75kmph respectively. It has been observed that pressure increases as the speed increases and higher values of pressure was observed at a speed of 75 kmph. Also the amount of pressure was reduced for the proposed model when compared with existing model at all speeds.

3.6 VELOCITY VARIATIONS AT DIFFERENT SPEEDS

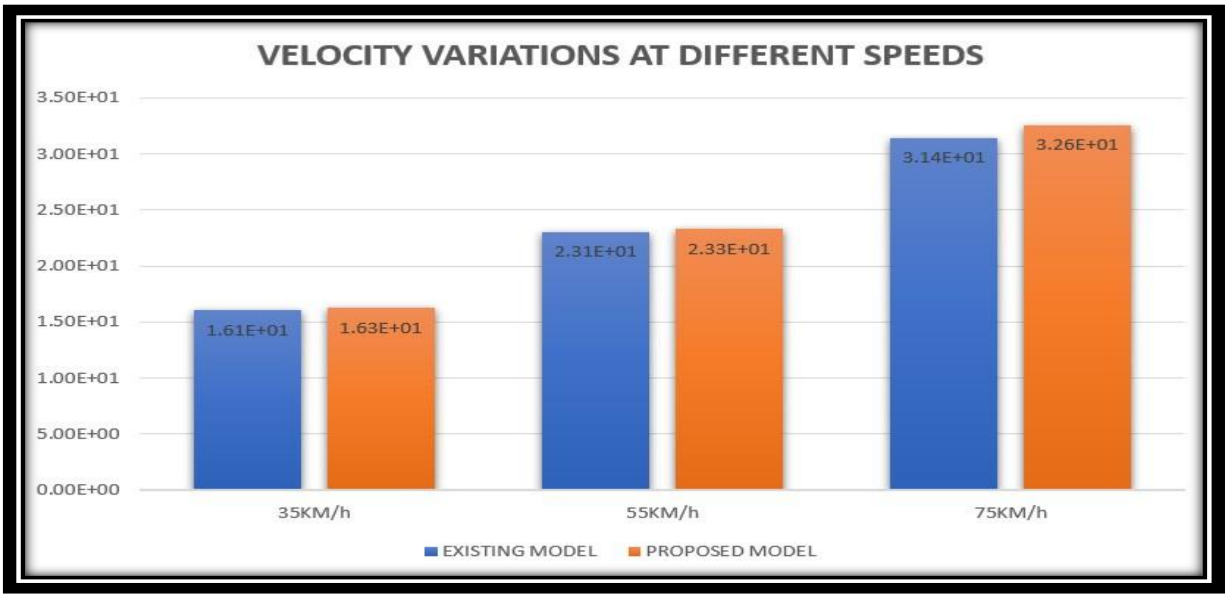


Fig.3.10: Velocity Variations at Different Speeds

Fig.3.10 shows the velocity variations for two models of trucks i.e., existing and proposed at speeds of 35kmph/55kmph/75kmph respectively. It has been observed that velocity increases as the speed increases and higher values of velocity was observed at a speed of 75 kmph for both the trucks. Also the amount of velocity was slightly increased for the proposed model when compared with the existing model at all speeds.

3.7 CALCULATION OF DRAG COEFFICIENT AND FUEL CONSUMPTION:

DRAG FORCE (N)

Drag force values are calculated for both the models of vehicle truck at different speeds 35kmph, 55kmph and 75 kmph using the following formula and are tabulated in the Table 3.1.

$$F_{\text{drag}} = \frac{1}{2} \cdot C_d \cdot \rho \cdot A \cdot V^2$$

Where:

- Fdrag= Drag force (in Newton, N)
- Cd = Drag coefficient (dimensionless) – This depends on the shape of the object and flow conditions.
- ρ = Fluid density (in kg/m³) – For air at sea level, ρ is approximately 1.225 kg/m³.

- A = Reference area (in m^2) – Usually the frontal area of the object that faces the fluid flow.
- V = Velocity of the object relative to the fluid (in m/s).

Table 3.1: Drag force values for both the models of vehicle truck at different speeds

Speed in Km/h	Velocity in m/s	Drag force for basic (N)	Drag force for Proposed (N)
35	9.722	1121.39	513.7
55	15.277	2121.57	836.42
75	20.833	3860.76	2982.1

DRAG COEFFICIENT

Drag coefficient values are calculated for basic and proposed models of vehicle truck at a speed of 75 kmph using the following formula.

$$\boxed{C_d = \frac{2 \cdot F_{\text{drag}}}{\rho \cdot A \cdot V^2}}$$

Where .C_d – Drag coefficient,
ρ - Air density
A – Frontal area = (A)
V – Velocity in X direction

The drag coefficient values for basic and proposed models are 0.4 and 0.3 respectively at a given speed of 75kmph and are given in the Fig.3.11. From the figure, it was observed that coefficient of drag was lower for proposed model of vehicle truck.

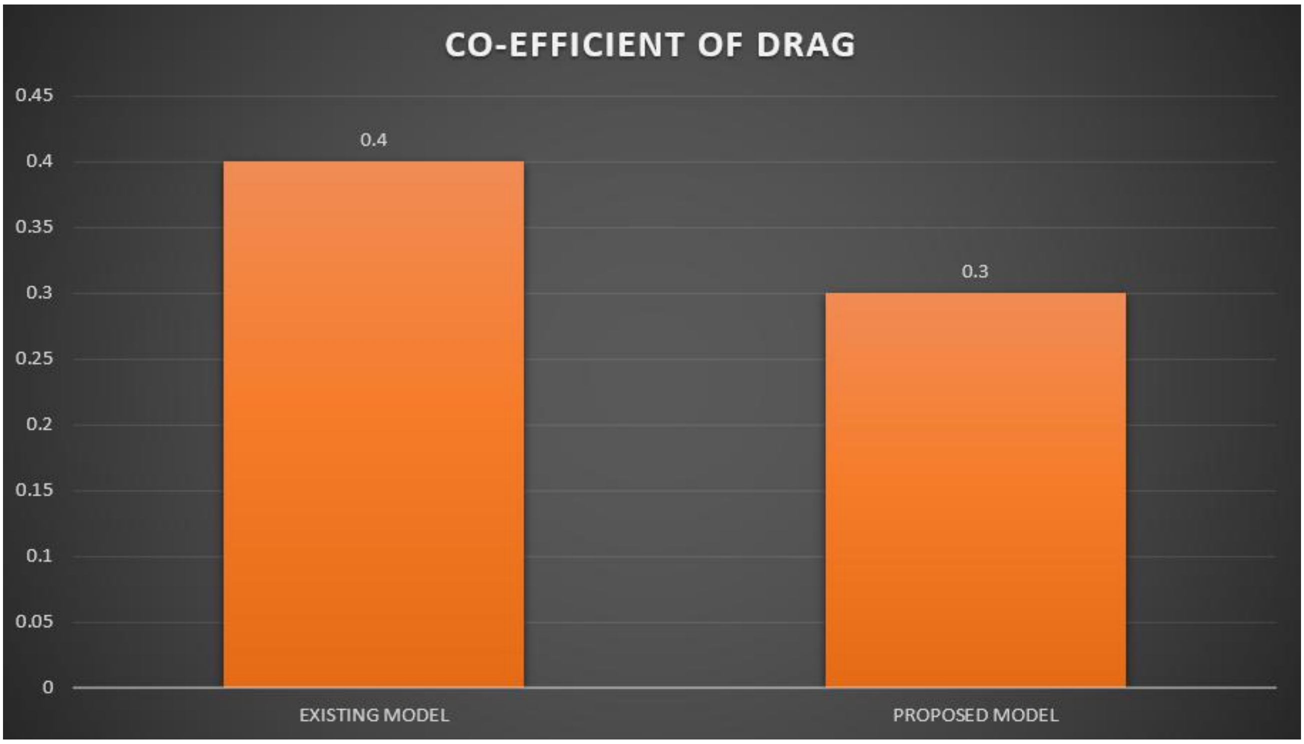


Fig.3.11: CO-EFFICIENT OF DRAG FOR BOTH MODELS

FUEL CONSUMPTION: Fuel consumption in litres for a speed of 75 kmph is calculated for both the models of vehicle truck as follows.

$$\boxed{L = (0.008051 \times F_d)}$$

Where, L is fuel consumption in litters

F_d is aerodynamic drag force in N

- **FOR FUEL CONSUMPTION OF BASIC MODEL:**

  $L = 0.008051 * F_d$

  $L = 0.008051 * 3860.76$

  $= 31.08$ L

- **FOR FUEL CONSUMPTION OF PROPOSED MODEL:**

  $L = 0.008051 * F_d$

  $L = 0.008051 * 2982.1$

  $= 24.09$ L

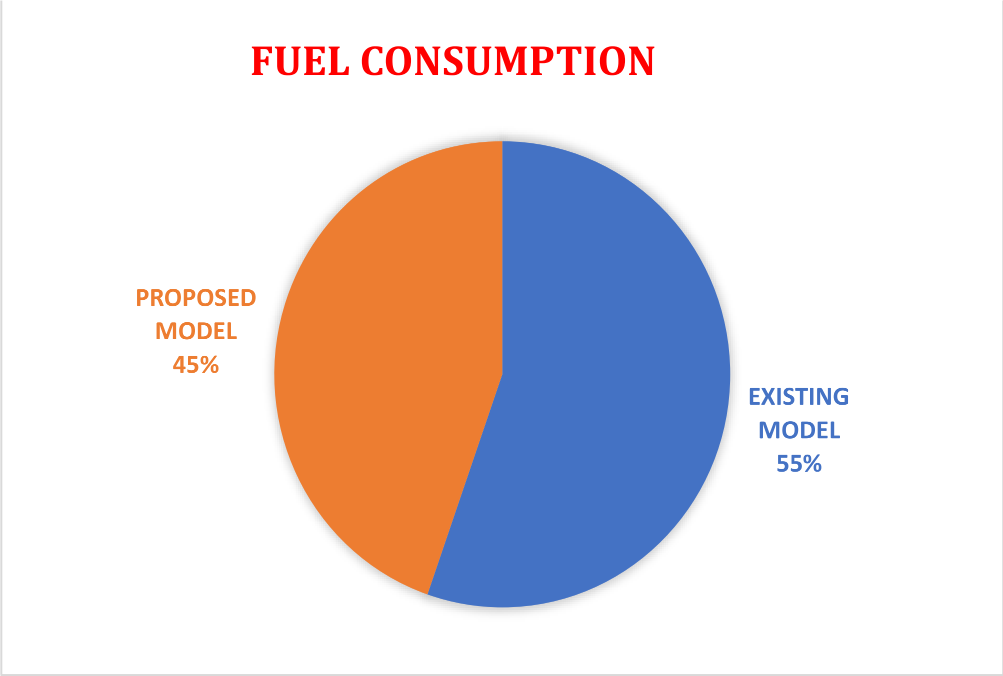


Fig.3.12: PERCENTAGE OF FUEL CONSUMPTION

From the Fig.3.12 it was concluded that fuel consumption is reduced for proposed Model when compared with existing model.

4 CONCLUSIONS

In this study, CFD analysis carried out on existing and proposed models of heavy vehicle truck to reduce drag and fuel consumption at different speeds of 35, 55 and 75 kilometres per hour. Analysis has been done for the proposed model by adding extra features like Deflector, Vanes between Cab & Container and Base Flaps at the end of Truck.

The following conclusions were drawn from this study.

(a) At a vehicle speed of 75 kmph, the drag coefficients for existing and proposed models are 0.4 and 0.3 respectively.

(b) Fuel consumption for existing model is 31.08 L where as fuel consumption for proposed model is 24.09 L at a vehicle speed of 75kmph.

From the above, it was concluded that the proposed model with features like deflector, vanes between cab & container and base flaps, giving lower values of drag coefficient and fuel consumption than that of existing model.

REFERENCES

1. Ananthan Raman,L., et.al., Methods for reducing Aerodynamic Drag in Vehicles and thus acquiring Fuel Economy, Journal of Advanced Engineering Research, ISSN:2393-8447, Volume 3, Issue 1, pp.26-32,2016.
2. Abdellah Ait Moussa, et.al., Aerodynamic Drag Reduction for a Generic Truck Using Geometrically Optimized Rear Cabin Bumps, Hindawi Publishing Corporation Journal of Engineering, Volume 2015, Article ID 789475, 2015.
3. Shobit Sengar, et.al., A Review paper on Aerodynamic Drag Reduction and CFD Analysis of Vehicles, International Research Journal of Engineering and Technology (IRJET), e-ISSN: 2395-0056, Volume 06, Issue 01, Jan 2019.
4. Abdul Kareem, et al., Calculation and optimization of the aerodynamic drag of an open-wheel race car, Journal of Engineering Science and Technology, Eureca 2013, Special Issue,1-15,2014.
5. J. Abhinesh et al., CFD analysis of aerodynamic drag reduction and improve fuel economy, International Journal of Mechanical Engineering and Robotic Research, ISSN: 2278 – 0149, Vol. 3, Issue 4, October, 2014.
6. Ashfaque et al., Drag Force Analysis of Car by Using Low Speed Wind Tunnel, International Journal of Engineering Research and Reviews, ISSN 2348-697X (Online), Vol. 2, Issue 4, pp.144-149, 2014.
7. Francesco mariani et al., A review paper on aerodynamic drag reduction and cfd analysis of vehicles, International Journal of progressive research in engineering management and science, vol. 03, issue 04, pp. 966-970,2023..
8. Upendra S. Rohatgi et al., Methods of Reducing Vehicle Aerodynamic Drag, Presented at the ASME 2012 Summer Heat Transfer Conference Puerto Rico, USA, July 8-12, 2012.
9. Yingchao Zhang et al., Aerodynamic Drag Reduction for a Generic Truck Using Geometrically Optimized Rear Cabin Bumps, Hindawi Publishing Corporation Journal of Engineering, Volume 2015, Article ID 789475,2015.
10. Keisuke nisugi et al., An Overview of Aerodynamic Drag Reduction and Vehicle CFD Analysis, Juni Khyat, ISSN: 2278-4632, Vol.10, Issue.2, 2004.
11. Rose mccallan et al., A Review paper on Aerodynamic Drag Reduction and CFD Analysis of Vehicles, International Research Journal of Engineering and Technology, (IRJET) e-ISSN: 2395-0056, Vol. 06, Issue. 01, Jan 2019.
12. R.H. Heald., A review paper on aerodynamic drag reduction and cfd analysis of vehicles, International Journal of progressive research in engineering management and science, vol. 03, issue 04, pp. 966-970,. April 2023.