Optimization of Truck Cabin and Trailer Shape to Reduce Aerodynamic Losses

[1]Desai Jay, [2]Imran Molvi
[1]M. Tech Student, [2]Assistant Professor
[1,2] Department of Mechanical Engineering,
[1,2] Parul Institute of Engineering and Technology, Parul University, Vadodara, India.

Abstract: Nowadays the reduction of drag is becoming a very important challenge for all the trucks manufacturers as they are competing to produce powerful pickup cars with the better gas mileage in the market regulated with low fuel emissions and consumer's need for bigger size trucks with more horse power and cargo capacity. Lower drag provides better performances such as higher top speed and better stability. It also often lower aerodynamic noise and greenhouse gas emission above all decreases in fuel consumption. The purpose of this research is to design truck cabin and trailer to reduce aerodynamic drag of the vehicle. The research approach is using computational fluid dynamics (CFD) technique. However, modern designs of pickup trucks tend to go higher and wider and thus they have higher frontal areas due to the functional, economic and aesthetic requirements. Increasing frontal area of the vehicle tend to increase the drag force acting on the vehicle which is proportional to the dimensionless drag coefficient (C_d) and the projected area of the vehicle. Consequently, to hold or even decrease the drag on a truck that has a larger frontal area, tremendous effort has to be made.

***Keywords* - Aerodynamic drag, truck-trailer, ANSYS fluent, profile modification, CFD, wind tunnel.**

I. INTRODUCTION

The continuously increasing fuel price has created widespread interest in vehicle with high efficiency including trucks, vans, SUVs. According to International Council on Clean Transportation (ICCT) India's diesel consumption has doubled in the past decades, increasing from 36.6 million metric tonnes in 2002 to 72.9 MMT in 2015. The regulations are aimed to reduce fuel consumption and greenhouse- gas emission from diesel powered truck and buses with a gross vehicle weight of 12 tonnes or greater.

Today auto manufacturers are producing powerful pickup car with better gas mileage in the market regulated with low fuel emission and consumer's need for bigger size truck with more horse power and cargo capacity. Efficiency of vehicle can be improved by reducing the total structural mass, or alternating the exterior body shape to reduce the aerodynamic drag. Therefore, improving vehicle aerodynamic is one of the factors that play crucial role for getting better mileage and better performance including the handling of the vehicle especially at high speed. [1-7]

II. DESIGN OF TRUCK

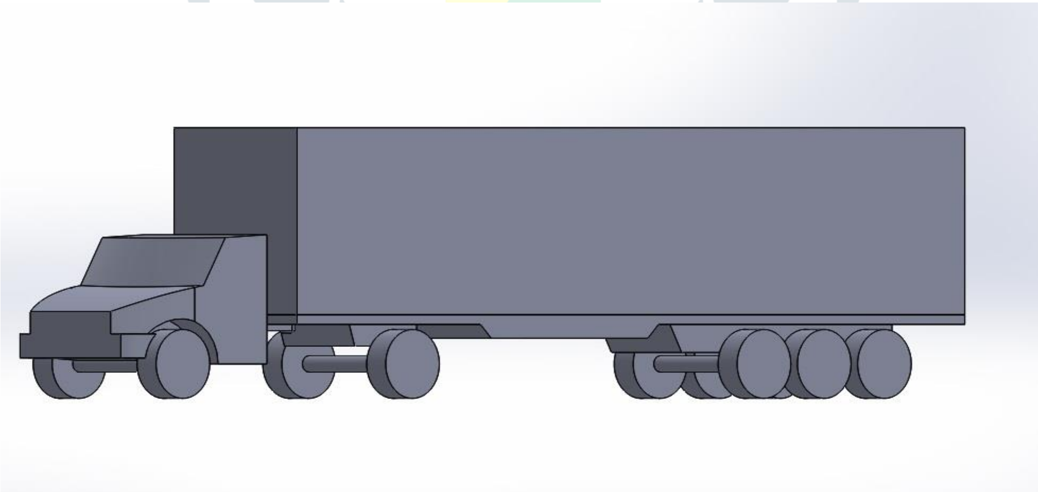

Fig.1 Design of Base Model of Truck

III. TECHNICAL SPECIFICATION OF TRUCK

Engine	: Detroit[tm]DD15
Power	: 350-600 HP
Torque	: 2779.42679 N-m
Transmission	: Manual Transmission
GVW	: 46 ton
GCW	: 105 ton
Wheel Base	: 8200 mm
Front Track	: 2450 mm

IV. DESIGN OF MODIFIED MODEL

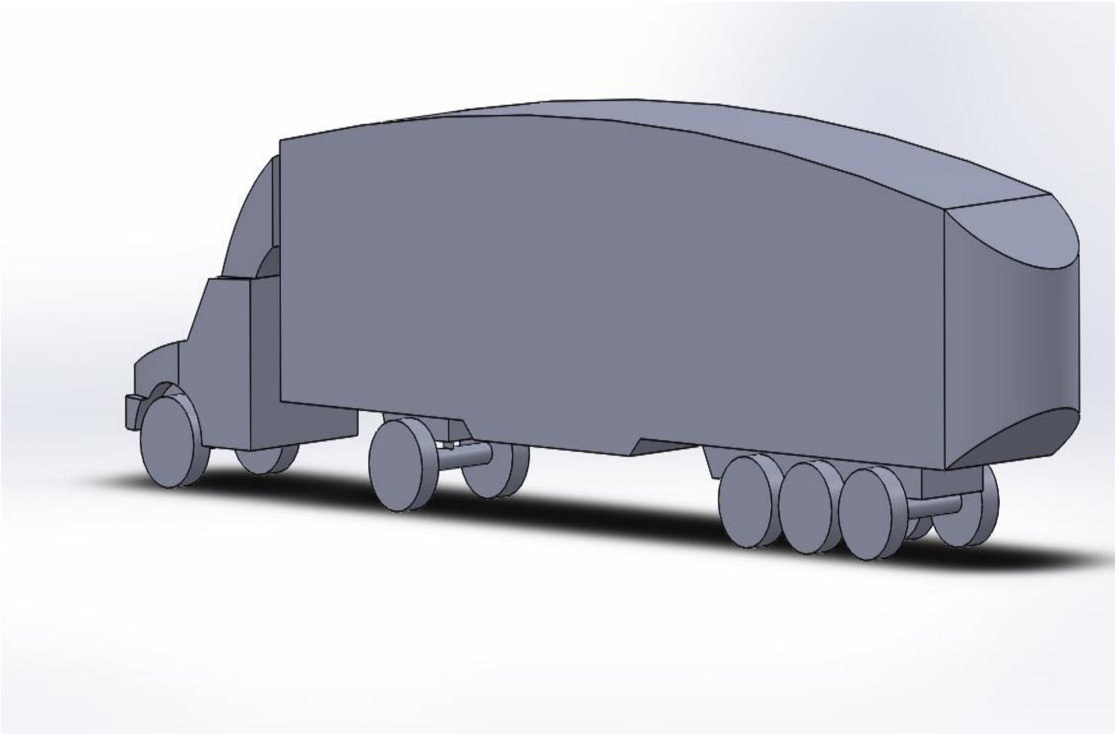

Fig. 2 Modified Designs 1 with Tail Angle 30°

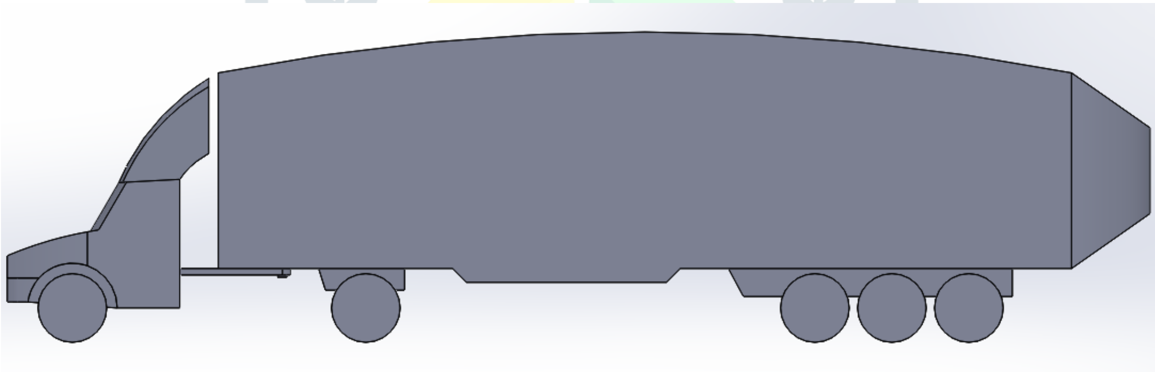

Fig. 3 Modified Design 2 With Tail Angle 35°

V. CALCULATION OF EQUATION

Drag Force : $\frac{1}{2}\rho V^2 A C_d$ (1)

Lift Force : $\frac{1}{2}\rho V^2 A C_l$ (2)

Where;

C_d	: Co-efficient of Drag
C_l	: Co-efficient of Lift
A	: Frontal Area of the Vehicle
ρ	: Air Density
V	: Vehicle Velocity

Fig. 4 3D Model of Modified Truck with Tail Angle 35°

VI. MESH GENERATION

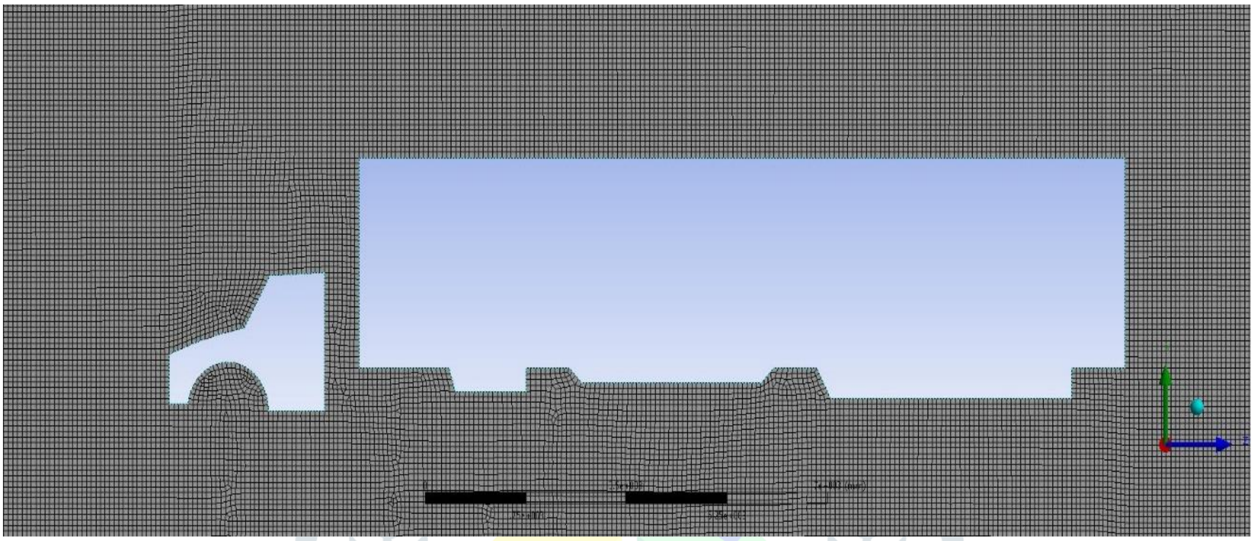

Fig. 5 Fine Mesh in Base Model

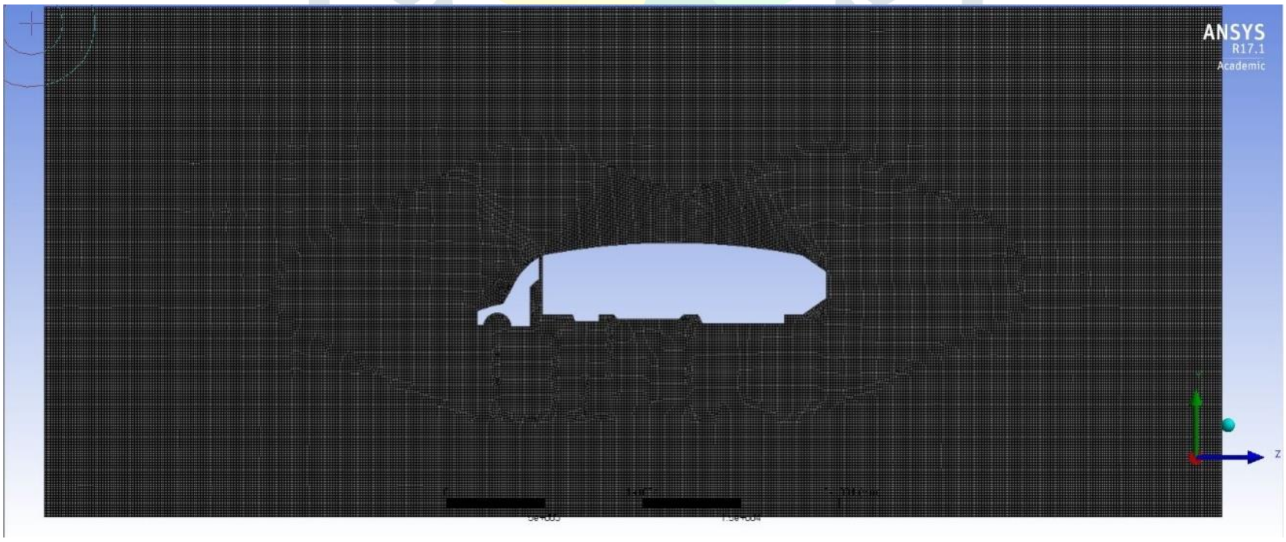

Fig. 6 Mesh in 35° Model

The triangular shape surface mesh was used due to its proximity to changing curves and bends. These elements easily adjust to the complex bodies used in automobile and aerospace bodies. With the default setting for mesh generation, ANSYS Meshing has generated the meshes. With the global mesh sizing settings, ANSYS Meshing recognized that there are some curvatures around the automobile body. But the meshing was very fine and it was only the initial guess by the software. The first things we changed the mesh size.

Table 1 Mesh Sizing Parameter

Global Mesh Sizing Setting	
Use Adaptive Size Function	Yes
Relevance Center	Coarse
Initial Size Speed	Active Assembly
Smoothing	Medium
Transition	Slow
Span Angle Center	18º
Proximity Accuracy	0.5
Maximum Size	80 mm
Minimum Size	47.630 mm
Growth Rate	1.2
Body Sizing	80 mm

VII. PRESSURE COUNTER

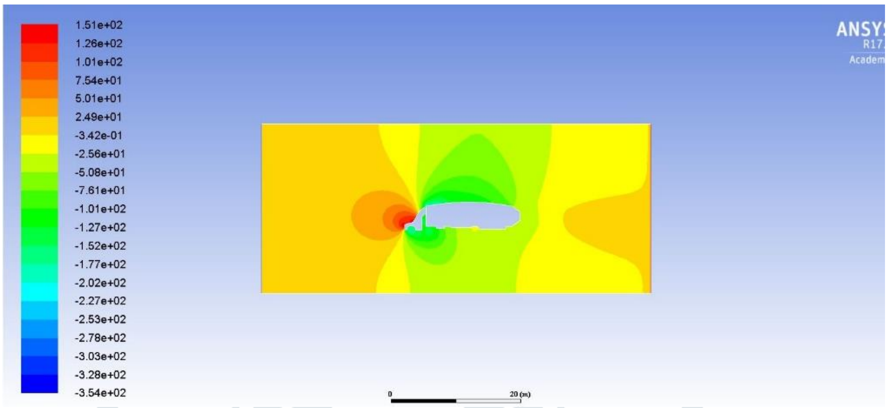

Fig. 7 Pressure Counter in 35° Model

VIII. VELOCITY COUNTER

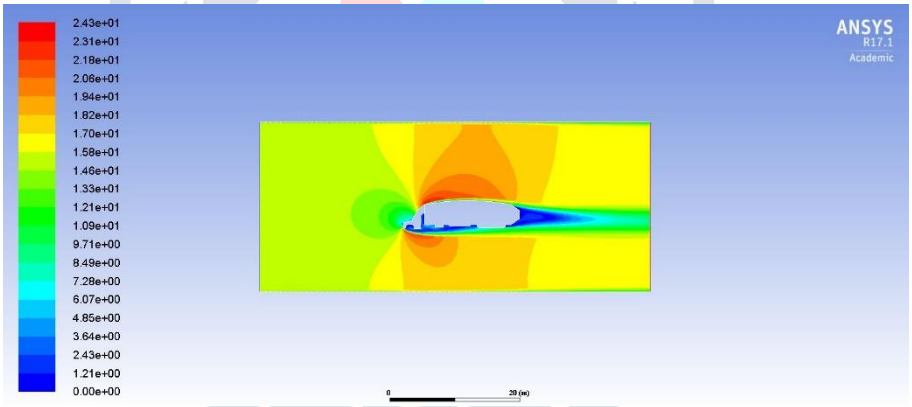

Fig. 8 Velocity Counter in 35° Model

IX. CO-EFFICIENT OF DRAG

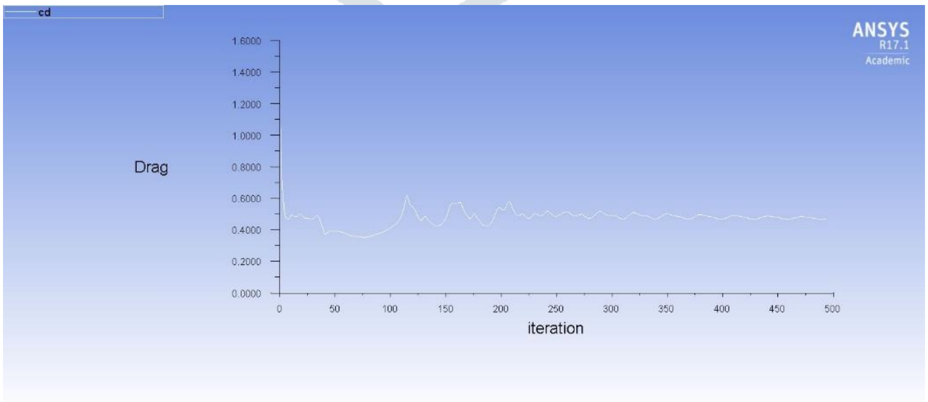

Fig. 9 Co-efficient of Drag in Base model

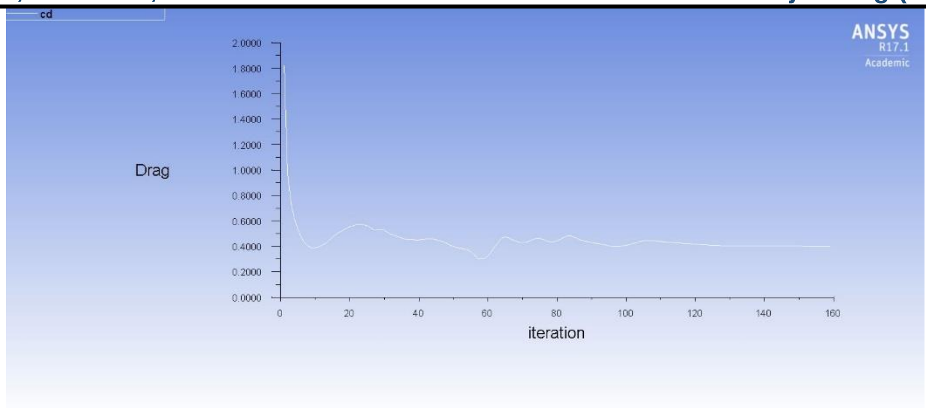

Fig. 10 Co-efficient of Drag in 35° model

X. RESULTS

Table 2 CFD Results

Velocity: 15m/s			
Sr. No.	**Model**	**Co-efficient of Drag (C_d)**	**Co-efficient of Lift (C_l)**
1	Base Model	0.53	0
2	30º Angle	0.4021	0
4	35º Angle	0.39	0
5	40º Angle	0.43	0

Table 3 Wind Tunnel Testing Results

Velocity (m/s)	Drag (N)	C_d	Lift (N)	C_l
17	4.8	0.377	13.7	0
34	5.9	0.3515	13.7	0

XI. CONCLUSIONS

The effects of different aerodynamics add-on devices on flow and its structure over a generic truck were analysed using CFD approach. The objective is to reduce aerodynamic drag acting on the vehicle and thus improve the fuel efficiency.

- The thesis studied the flows over a truck with add-on devices such as (1) Deflector, (2) Modified top side of the Trailer, (3) Rear Boat Tail at 3 different angles. All the studied add-on devices reduced the drag co-efficient when it compared to the result of base line truck.
- The maximum reduction of aerodynamic drag co-efficient(C_d) was 14.45% which was achieved by rear boat tail angle and it was followed by 35º angle.
- The SST transition model is used for maximum speed at 15m/s (55km/h). It is found that the drag co-efficient is reduced from 0.53 for base model and 0.39 foe modified truck model at 35º.
- However, other two modified model with angle 30º and 40º the co-efficient of drag value is increase 0.4021 and 0.422 respectively. That means as the rear boat tail angle is increased; simultaneously the value of co-efficient of drag is also increased.
- On the other hand, lift co-efficient is zero in both cases CFD analysis as well as Wind Tunnel Testing. Because there is no pressure difference in the top side and bottom side of truck.

XII. AKNOWLEDGEMENT

I acquire this opportunity with much pleasure to acknowledge my internal dissertation guide Mr. Imran Molvi, HOD, Mechanical Department PIET, Parul University and all the people who have helped me through the course of my journey.

REFERENCE

[1] Chaitanaya Chilbule, Awadhwsh Upadhyay, Yagna Mukkamala 2014. Analyzing the profile modification of truck-trailer to prune the aerodynamic drag and its repercussion on fuel consumption, 97:1208-1219

[2] Harun Chowdhury, HazimMoria, Abdulkadir Ali, Iftekhar Khan, Firoz Khan, Simon Watkins 2013.A study on aerodynamic drag of semi-trailer truck, 56: 201-205

[3] J. Bettle, A.G.L. Holloway, J.E.S. Venart 2003. A computational study of the aerodynamics forces acting on a tractor-trailer vehicle on a bridge in cross-wind. Journal of wind Engineering and Industrial Aerodynamics, 91:573-592

[4] Harun Chowdhury, Bavin Loganathan, IsratMustray, HazimMoria, Firoz Alam 2017. Effect of various deflectors on drag reduction for trucks,110:561-566

[5] R. Miralbes 2012. Analysis of some aerodynamics Improvements for semi-trailer tankers. World Congress on Engineering

[6] Subrata Roy, Pradeep Srinivasan. External flow analysis of a truck for drag reduction. SAE :2000-01-3500

[7] Richard M. Wood 2008. Operationally-practical & Aerodynamics-Robust heavy truck trailer drags reduction technology. SAE : 2008-01-2603

[8] Drew Landman, Richard Wood, Whitney Seay and John Bledsoe 2009. Understanding Practical heavy truck drag reduction limit. SAE: 2009-01-2890

[9] S.A. Coleman, C.J. Baker 1994. An experimental study of the aerodynamic behaviour of high-speed lorries in cross wind" Journal of Wind Engineering and Industrial Aerodynamics, 53: 401-429

[10] QI Xiao, LIU Yong-qi 2011. Experimental and Numerical studies of aerodynamics performance of trucks, 23(6): 752-758