

**Reduce Drag Force In Heavy Loaded Vehicles Using Software**

¹Sanjay H. Zala, ²Govind P. Chaudhari, ³Jayesh C. Dhandhalya,
⁴Darshan R. Dhandhaliya, ⁵Vipul J. Baraiya

Mechanical Engineering ,Government Engineering College, Bhavnagar

Abstract — One of the most important environmental issues within the automotive industry today is to reduce the fuel consumption and emissions. Together with the increased fuel price this has resulted in a “green race” within automotive companies in order to stay competitive, and the development of fuel efficient products has escalated. This is, among other methods, achieved by improving efficiency of the engine, reducing rolling resistance and improving aerodynamics. A recent project called Optic Fuel, conducted by Renault Trucks, contains an investigation of new technologies to reduce the fuel consumption. The report states that aerodynamic improvements of the tractor unit and the semi-trailer, and thereby drag reduction, is one of the most important issues when it comes to fuel saving.

Keywords- COMSOL 6.0, Tractor-Trailer Body, Modeling, Material Defined, Meshing, Boundary conditions.

1. INTRODUCTION

Aerodynamic is a study about air flow around an object. In land vehicle application, aerodynamic is one of land vehicle important design after engine and suspension. A smooth design is needed to reduce air resistance to the vehicle at high speed, because air resistance will increase as the velocity increase. The Computational Fluid Dynamic (CFD) is another method to study about the air flow around the vehicle. This method was used to give the users an overview of the air flow and expected result in the aerodynamic simulation. Hence, after the experiment in the wind tunnel, the users may verify they experimental result with CFD. In this project, analysis of aerodynamic will be done on a heavy truck by using CFD-commercial software. Objective of project is to suggest appropriate modification which helps to reduced aerodynamic drag. The required power to overcome aerodynamic drag and rolling resistance as a function of speed for a truck of 40 tons. As seen in the picture, there is a cubic increase of drag, whereas the rolling resistance grows linearly with speed. Above approximately 80 km/h the aerodynamic drag becomes dominating, which makes aerodynamics very important for long-distance transport where speeds up to 90 km/h is common. However, it should be mentioned that aerodynamic drag is still important below this speed, although not to the same extent. By reducing drag, less power is needed and fuel consumption can be reduced.

2. PROBLEM SUMMARY

Today's demand of reducing the fuel consumption of vehicles is one of the most challenging issues within the automotive industry. Together with the increased fuel price, the development of more fuel efficient vehicles has escalated. A recent research about fuel reduction technologies for trucks showed that aerodynamic improvement is one of the most important technologies when it comes to fuel saving. Trucks has a well established aerodynamic focus in the product development process of the tractor, although there are no focus on trailer aerodynamics. Therefore, more research of aerodynamics around the tractor and the trailer is desirable to see the possibilities to further reduce drag.

3. Basic of CFD

Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows. Mathematical modeling (partial differential equations), numerical methods (Discretization and solution techniques) Software tools (solvers, pre- and post processing utilities. CFD enables scientists and engineers to perform ‘numerical experiments’ The branch of mathematics that deals with the development and use of numerical methods for solving problems. Fluid mechanics is a branch of physics concerned with the mechanics of fluids (liquids, gases, and plasmas) and the forces on them. Fluid mechanics has a wide range of applications, including for mechanical engineering, civil engineering, chemical engineering, geophysics, astrophysics, and biology.

4. Tractor-trailer gap

The effect of the gap clearance between the tractor and the trailer is very much dependent on how large the gap is. One of the drag contributing issues with the larger gaps is that air goes into the gap and hits the trailer front, which results in an increased pressure drag. This pressure becomes even greater if there is a height difference between the tractor and the trailer. If this is the situation it is beneficial to use a roof deflector with an angle adjusted to the height of the trailer. During cross-wind conditions more air is entering the gap which will increase the flow separation, and thereby turbulence, on the leeward side of the trailer. This will affect the drag and stability. In order to reduce the drag due to the gap clearance there exist several different solutions of aerodynamic devices which can be implemented on the truck. Most common is the roof deflector and the side deflectors added on the tractor to guide the air over the gap. For those configurations where the gap is large, there are a number of devices to improve the aerodynamics. The main purpose of such devices, added on the trailer front, is to prevent uncontrolled circulation and cross-flow in the gap, improve the flow over the gap and to reduce the pressure acting on the trailer. Cross-flow vortex trap and Nose cone, are two examples of how aerodynamic trailer devices can be implemented in the gap. The Cross-flow vortex trap device is aiming at creating a couple of stable vortices at the trailer front in order to reduce the pressure in this region. The Nose cone is used to improve the flow over the gap and to reduce the pressure on the trailer front. Full-scale wind-tunnel tests conducted at the National Research Council of Canada (NRC), performed with a speed of 100 km/h and over a yaw sweep to calculate the wind average, states that these devices reduce drag by 2 and 34 drag counts, respectively .

5. Rear of the trailer

At the rear of the trailer a dominant base wake is created, containing unsteady turbulent flow. This is a result of a low-momentum flow along the top and the sides of the trailer that separates at the trailing edge of the trailer. The low-momentum flow from the undercarriage interacts with the base wake, resulting in an even greater turbulent base flow. A consequence of this is a low-pressure region that is created behind the trailer that contributes to drag. There exist different types of aerodynamic devices to decrease and stabilize the base wake by guiding the flow at the rear of the trailer. By extending the rear with different types of angled plates, the flow attachment is maintained and the air can be guided into the center of the base wake. Other devices are added to the sides and the roof of the trailer to generate energized vortices to delay the separation and thereby decrease the base wake. Base plates and Boat tail, see Figure 4 demonstrate how base treatment devices can be used at the trailer back in order to reduce the drag from the base region. According to wind-tunnel tests, these devices improve drag by 51 and 44 drag counts, respectively

6. Fluid dynamics

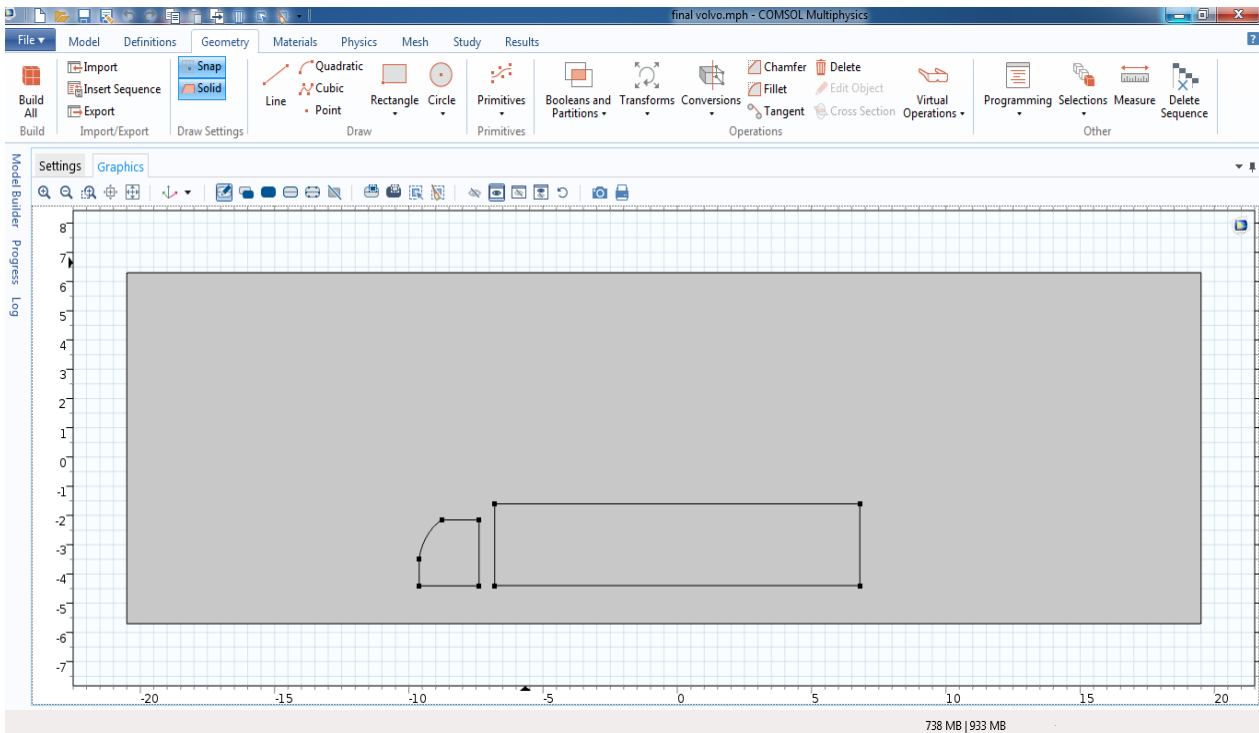
Fluid dynamics is the study of fluids in motion and the physics can be described by three conservation laws :

- (1) Conservation of mass - the continuity equation states that the amount of mass flow that enters a control volume must be equal to the amount leaving it.
- (2) Conservation of Linear momentum (Newton's Second Law of Motion) - given a Newtonian fluid, these equations are used to obtain a relation between pressure, momentum and viscous forces, the equations are called Navies-Stokes equations.
- (3) Conservation of energy (First Law of Thermodynamics) - the energy equation is the law saying that the total amount of energy is conserved within the system, however, it can change between the different states.

7. IMPLEMENTATIONS & DESIGN

1. Modeling

Motion	:Stationary
Time	:Steady
Over all length	:15835 mm
Over all width	:2266mm
Wheelbase	:3600mm
Rear overhang	:2580mm
Front overhang	:1200mm
Initial height	:2895mm



2. Material Define

Solution method	Selection
Space	Two dimension
Motion	Stationary
Time	Steady
Fluid material	Air & Gas
Flow of Fluid	Segregated Flow solves the pressure and velocity Equations
Equation of state	Constant Density
Viscous model	k-epsilon turbulence model

3. Meshing

The computational domain is designed to lead to a free with neglect able blockage, which essentially means a box that consists of an inlet, an outlet, two sides, a roof and a ground Surface. The size of the domain is taken approximately such that the real-time road conditions were satisfied. The surface mesh was created on the geometry of the vehicle as well as on the surface of the domain. Between the surface of the vehicle and the domain the computational grid was generated. To capture certain areas of interest (where separation might occur and where the degree if turbulence is high) the cells have to small enough to solve all irregularities and achieve a robust solution. The grid has been redefined around the bus, at the rear and especially underneath the vehicle since this study is focusing on the underbodies influence on the flow field. In previous sections it has been mentioned how the air act when flowing past a surface and create a boundary layer. To capture these floe phenomena refinement region around the exposed surface must be created, this refinement area is called a prism layer.It's important to define the orientations of the surfaces in order to make the prism layer to grow in the direction of normal vectors. The prismatic layer is needed to predict the flow more accurate since the highest gradients are located near the wall

4.Boundary Condition

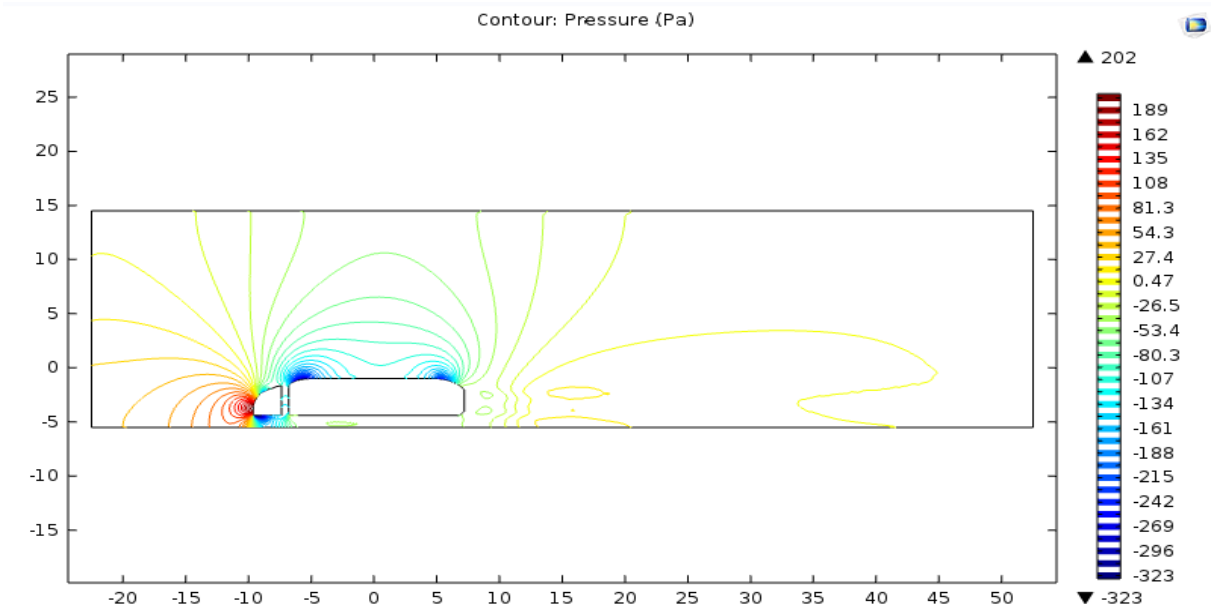
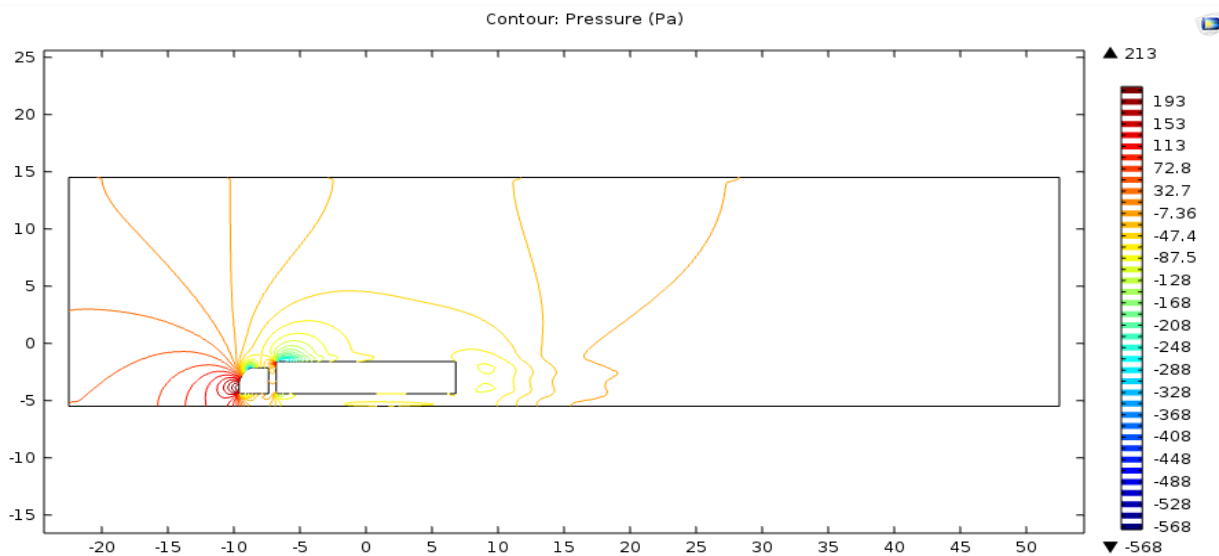
Boundary conditions were applied on the meshed model. In the simulation only straight wind condition was considered at different vehicle speeds up to 100 Km/hr. Constant velocity inlet condition was applied at the inlet to replicate the constant wind velocity conditions same as wind tunnel tests. Zero gauge pressure was applied at the outlet with operating pressure as atmospheric pressure.

Boundary	Name	Boundary Condition
Left Edge	Inlet	Velocity Inlet
Right Edge	Outlet	Pressure Outlet
Top & Bottom Edge	Wall	Wall
Truck Profile	Body wall	Wall

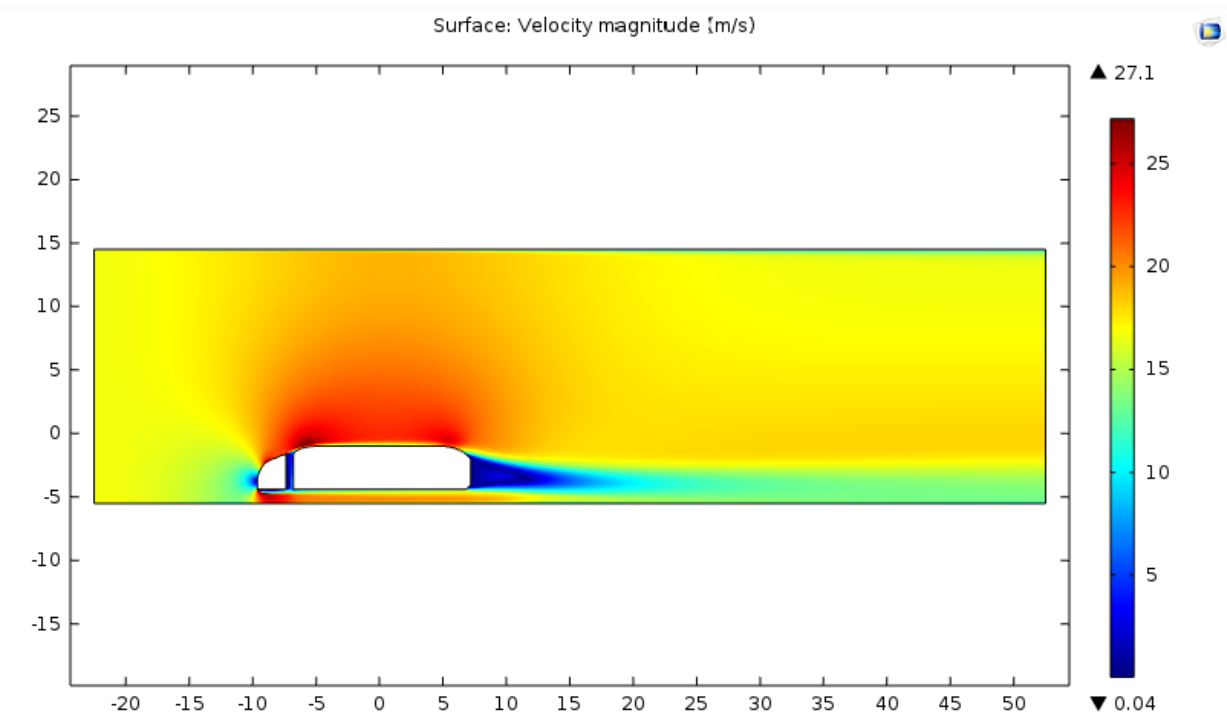
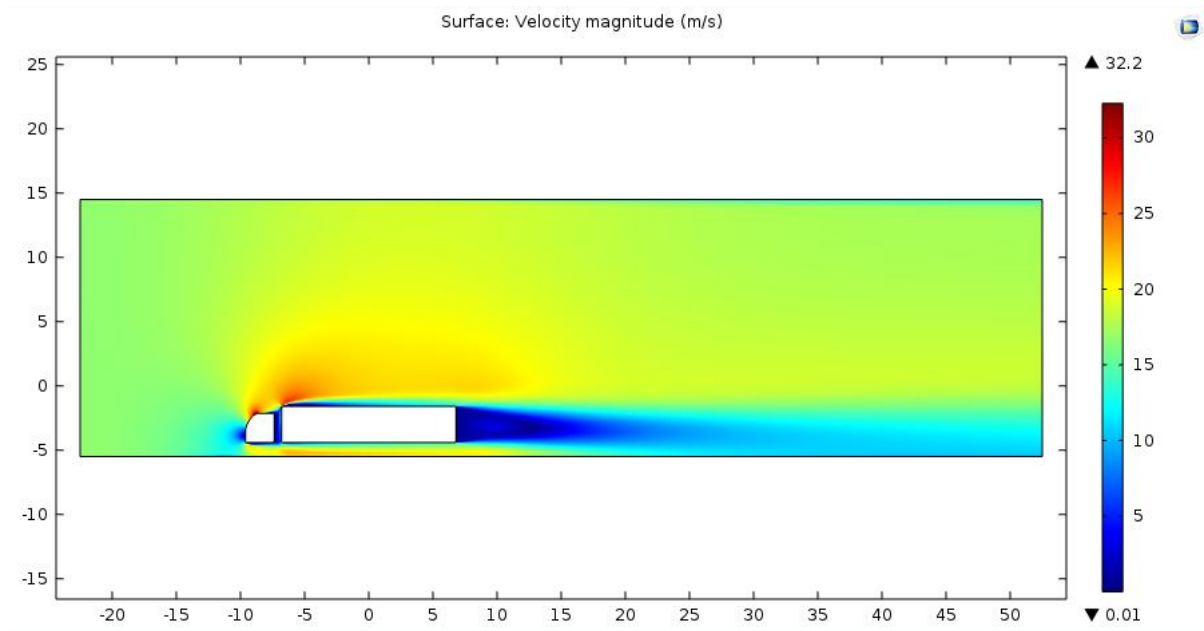
8. COMPARISION

In our case we find out coefficient of drag value (C_d). And we got 0.6579 for our case. And for original value for ahemad body(0.8).So we can say that drag force reduces with help of change in design.

Pressure diagram



Velocity diagram



9. CONCLUSIONS

This thesis verifies the possibilities of improving the aerodynamics around a truck in order to reduce the fuel consumption, and concludes that:

Aerodynamic trailer devices have a great potential of reducing drag. Compared to the tractor, the trailer is much more susceptible for aerodynamic drag improvements and thus the fuel consumption can be substantially reduced by using trailer devices. By combining the devices, even larger drag improvements can be achieved. The undercarriage and the base of the truck are the two regions where the greatest effects are achieved when adding aerodynamic devices to the trailer. The devices Side skirts and Frame extension have during this project shown a large potential to

improve the flow in these regions and should be of great interest for further development. It is not beneficial to aerodynamically change the geometry of the trailer since there are relative simple add-on devices that shows the equivalent effect. As mentioned in Chapter 1, the tractor at Mahindra already has a relatively good aerodynamic shape and adjustments are limited. Therefore, to achieve further aerodynamic improvements, and thereby reduce the fuel consumption and emissions, the next step for these companies should be to consider the whole truck during the aerodynamic development. In order to do this, a co-operation between the tractor and trailer manufacturers is recommended and communication between these two should be established. The advantage if the tractor and the trailer were to be developed together especially applies to the interface between the cab and the trailer front, but also between the chassis and trailer underbody. A mutual development would make it possible to optimize the integration of these components, which would improve the flow transition and thereby improve both the undercarriage flow and the base flow.

10. REFERENCES

- 1.CFD Module User's Guide (2011)
- 2.Frank M. White. Fluid Mechanics.
- 3.Meshing Research Corner. <http://www.andrew.cmu.edu>
- 4.L. Davidson. An introduction to turbulence models.
- 5.<https://www.quora.com/What-is-the-scope-for-CFD-in-India>.
- 6.Volvo Trucks global homepage. www.volvo/trucks/global
- 7.SachinThorat and G Amba Prasad Rao (1999), "Conducted an Research on Computational Analysis of Intercity Bus with Improved Asthetics and Aerodynamic Performance on Indian Roads".
- 8.Edwin J Saltzman, Robert R Meyer, McCallen and K Salari (2007), "InternationalPapers Carried Out Studies on Reducing 440 Int. J. Mech. Eng. & Rob. Res. 2014 J Abinesh and J Arunkumar, 2014 the Drag of Trucks and Buses".
- 9.Peterbilt (2009), "Motors Company Presents a White Paper on Heavy Vehicle Aerodynamics and Fuel Efficiency".
- 10.McCallen (2004), "Investigated on a Reduction in Drag Value Until the Front Leading Edge".
11. G Buresti, G V Lungo and G Lombardi (2007), "Carried out a Research on Methods for the Drag Reduction of Bluff Bodies and Their Application to Heavy Road".