TEFUGEN
Technologies for Future Generation

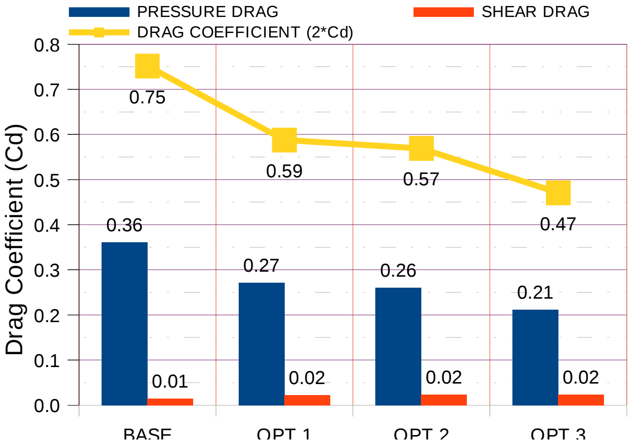
EXTERNAL AERODYNAMIC CFD ANALYSIS – DRAG ON A TRUCK

Objective

To analyze the external aerodynamic characteristics of a Commerical road vehicle (Truck) and identify optimal external shape required to reduce the drag force without altering the other essential aerodynamic characteristics of the truck.

Challenges

- Suitable external shape modifications in the critical areas that alters the drag co-efficient
- Suitable meshing scheme and flow model required to predict the flow characteristics accurately
- Capturing complex geometrical features in areas of interest during mesh generation



Graph 1 – Drag Coefficients of Truck

CFD Model

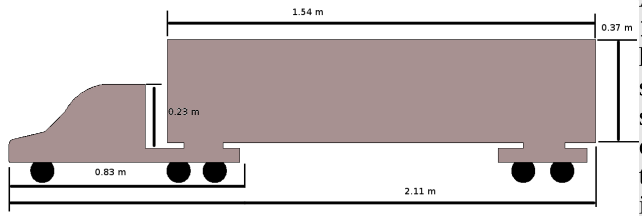


Fig-1 Geometry – Trailer Truck

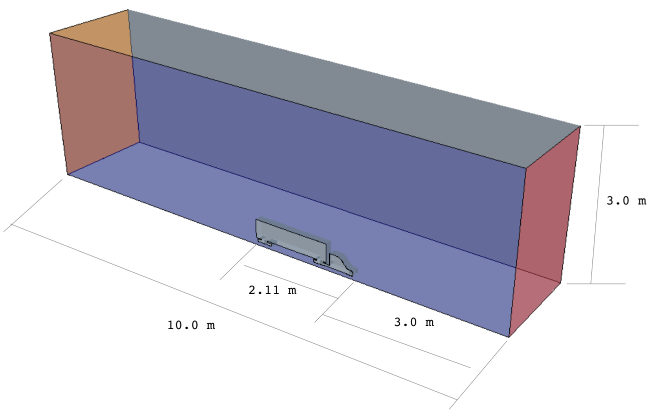


Fig-2 Domain – Trailer Truck

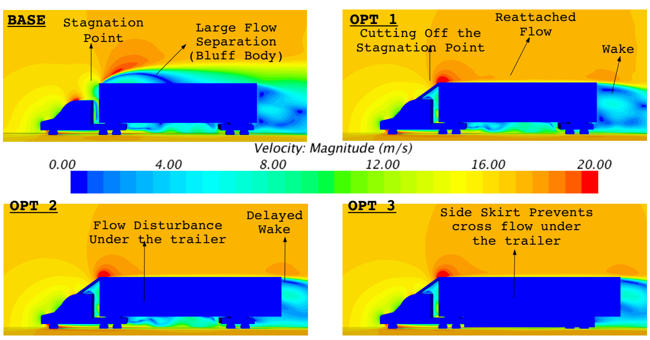


Fig-3 Velocity Contours

Approach

A scaled wind tunnel model of a typical trailer truck **(Fig-1)** was considered for the study. These are bluff bodies with high drag coefficients due to their shape. Steady state solver with RANS K-omega turbulence model was used to simulate the flow and predict various aerodynamic force coefficients at a velocity of **60 Kmph**.The choice of turbulence model depends on the level of accuracy required in the predicted results.External shape optimization was then completed through suitable geometrical modifications.

Conclusion

The optimization was carried out to compare the flow behaviour around the truck with geometry change. There was a reduction of **37.40%** in the **drag coefficient** between the initial and final design.

Sport utility vehicles, trucks, vans and buses are examples of large ground vehicles that are often criticized for their poor fuel economy. With recent surge in fuel prices and stringent norms for reducing emissions, automotive design engineers are being challenged with an immediate task of developing more fuel efficient vehicles with better aerodynamic designs. Through CFD, problem areas like recirculation zones, flow separations can be identified easily and design optimizations can be done in short durations. For a full sized truck, **"reduction of drag co-efficient (Cd) by a value of 0.01 will approximately yield an increase of 0.1 mile per U.S gallon in mileage" [1]**

Benefits

- Detailed analysis of flow behaviour at the wake
- Identifying critical flow separation areas that contributes to increase in drag force
- Various flow conditions like side-slip and various cross winds at different angle of attack can be simulated

Applications

- External shape optimization of large commercial vehicles and passenger cars
- Stability analysis of vehicles in various conditions

1) www.edmunds.com/fuel-economy

COMPUTATIONAL FLUID DYNAMICS
TEFUGEN TECHNOLOGIES PRIVATE LIMITED
G Floor, L-2, Electrical & Electronics Industrial Estate, Thuvakudy, Tiruchirapalli – 620 015. Tamilnadu, India.
+91 431 2500322 +91 431 2501134 tefugen@tefugen.com www.tefugen.com