EXTERNAL AERODYNAMIC ANALYSIS OF BUS

Objective

To assess and compare the aerodynamic drag and lift force generated in light and medium duty commercial vehicles (Buses) with respect to their external body shape.

Challenges

- Geometry clean up and discretization.
- Choosing suitable meshing scheme and flow model.
- Capturing complex geometrical features in areas of interest during mesh generation

CFD Model

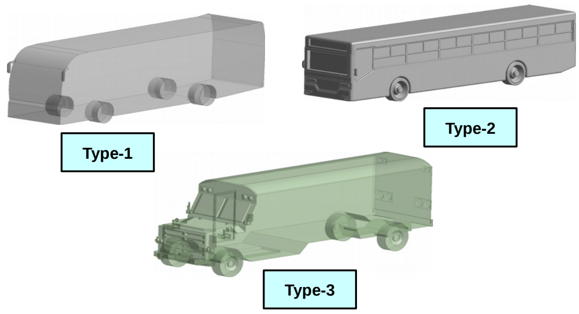

Fig 1: Bus geometry

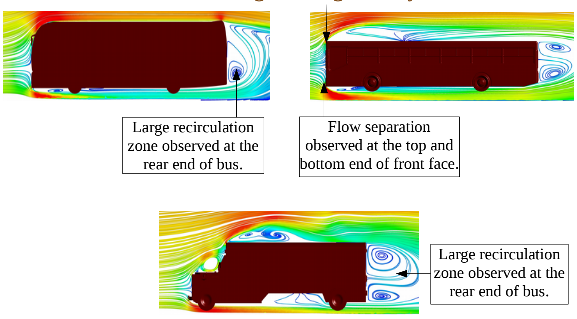

Fig 2: Velocity Streamlines

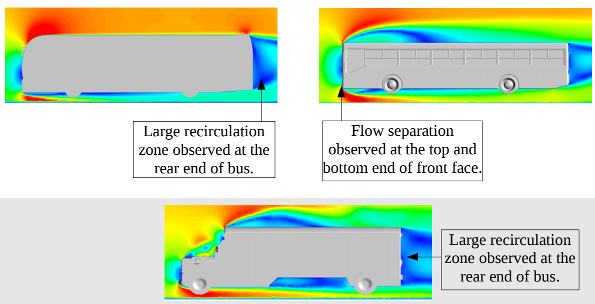

Fig 3: Velocity Contours

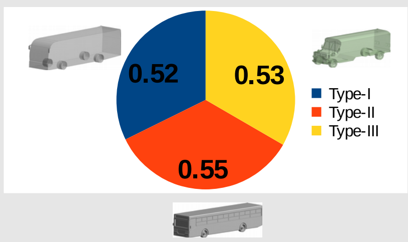

Fig 4: Comparison of Drag Coefficient

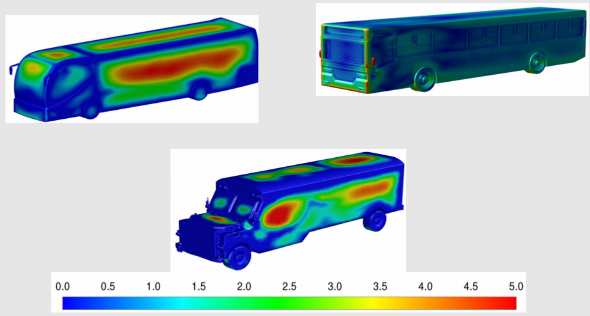

Fig 5: Wall y+ Contour

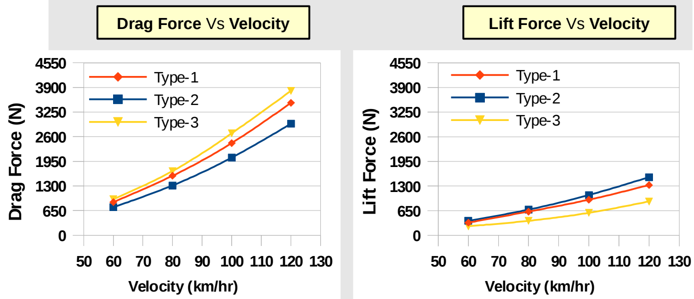

Fig 6: Force vs Velocity plots

Approach

Reducing fuel consumption and minimizing losses caused due to aerodynamic forces (drag & lift) is one of the primary focus while designing external shape of any commercial vehicle. In this case study, effect of vehicle's frontal shape to the drag and lift force generated were investigated using CFD technique. Commercial Vehicles with three different frontal shapes were considered **(Fig 1)** for investigation. Flow physics over the vehicle's body were resolved using **RANS based SST-K-ω,** near wall physics were predicted using finer mesh near the wall with y+ values ≃1 **(Fig 4)**. The drag and lift forces generated when vehicles travel at different speed were predicted and compared; drag vs velocity plots were generated **(Fig 5)**. Formation of wake region behind the vehicles were assessed in detail and velocity streamlines were generated for better visualization of flow separation and wake formation zones within the vicinity of the vehicle.

Conclusion

Results obtained were compared to gain better insight into the effect of frontal shape on the aerodynamic forces generated during vehicle movement and also to reduce the resulting inefficiency. The effect of vehicle speed on the drag force and wake region formation were also clearly elucidated. Suitable frontal area designs with low drag force and better manoeuvrability can be adopted to improve the efficiency of the vehicles.

Benefits

- Losses due to aerodynamic forces can be minimized.
- Reduction in drag also reduces fuel consumption.
- Design of external shape with better aesthetics and aerodynamic efficiency.
- Design of suitable drag reducers.

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 cae.info@tefugen.com www.tefugen.com