A CFD study of Wind Tunnel Wall Interference

Wael Mokhtar[*] and Md Rakibul Hasan[**]
School of Engineering
Grand Valley State University
Grand Rapids, MI 49504
Email: mokhtarw@gvsu.edu, hasanmd@mail.gvsu.edu

Abstract

Wall interference is one of the most crucial factors that creates inaccuracy for wind tunnel measurement which is an indispensable tool to predict the aerodynamic performance in automotive and aircraft industries. In recent years, there is an interest in reduction of drag coefficient for commercial high blockage vehicles like buses and semi-trucks. Flow over these vehicles is complex and difficult to predict with accuracy. In order to have efficient aerodynamic design, accurate data collection from wind tunnel is of great importance. The present work attempts to perform a Computational Fluid Dynamics (CFD) study of high blockage vehicle in order to assess the wall interference effect by analyzing the flow structures for different blockage ratios. The blockage ratios are generated by varying the areas of the wind tunnel cross section. It also focuses on use of CFD to calculate corresponding drag corrections on the basis of wall-signature method.

The study uses a generic model for a semi-truck in a virtual closed wall test section. For all the cases, the model size was constant and the blockage ratio was increased by changing the test section size.

The simulation was run for city speed (50 mph), high speed (70 mph) for each blockage ratios using the aforementioned Physics and Mesh settings. The stopping criteria of simulation was set as 500 iteration. Drag Coefficient for each case was calculated for the frontal area of 9.752396 square meter. The commercial CFD software STAR CCM+ was used in the study.

Introduction

Wall interference is one of the current challenges in wind tunnel testing of commercial high-blockage vehicle such as buses and semi-trucks. With the recent interest in drag reduction tools, the need for more accurate wind tunnel results increases every day. The classical method of images gives reasonable estimates of drag corrections. The use of wall-signature method improves significantly this method. Designers will therefore continue to rely on wind tunnel testing in their efforts to produce low-drag designs. Efforts like minimizing wind noise, optimizing engine cooling, minimizing wind effects on vehicle handling etc. will also depend on wind tunnel testing.

[*] Ph.D, Director of School of Engineering
[**] Graduate Student

In order to have efficient aerodynamic design for these kind of vehicles, accurate data collection from wind tunnel is of great importance. The wind tunnel flow does not usually corresponds to the free flow because of wall and mountings effect. So, aerodynamic performance data resulting from the wind tunnel experiment therefore are affected by certain systematic errors. In order to minimize these influences to a large extent, proper data corrections of the wind tunnel tests are necessary. Wind tunnel walls impose constrains on the flow around the model. Blockage ratio (ratio of the model frontal area to cross sectional area of wind tunnel test section) plays a vital role for automobile models to cause the wall interference effect. It has a significant effect on drag co-efficient as well.[1]

In the present work, a CFD study is presented to address two major aspects of the wall interference. The first focuses on the flow structures that contribute to the wall interference. The study performed a simulation of 53 feet full scale semi-truck model in order to assess the wall interference effect by analyzing the flow structures for two different blockage ratios. The blockage ratios are generated by varying the areas of the wind tunnel cross section. It also focused on use of CFD to calculate corresponding drag corrections on the basis of wall-signature method.[2]

Literature Reviews

A comprehensive study of wind tunnel wall interference effects and its correction procedures are done by Edwald et al.[3]. Maciejewski et al.[4] has reported the numerical simulation of the blockage effects in wind tunnel on automobile bodies using an incompressible Navier-Stokes solver and the effect of blockage on drag force is presented for different tunnel heights. Griffith et al.[5] gives computational 2-D results for semi-circular blockages for laminar flows with Reynolds number 50 to 3000 for small to large blockage ratio cases for very low speed. Amrouche et al.[6] give experimental effects of pressure distribution due to blockage condition for grid turbulent flows for prisms for different small blockage ratios. Saltzman and Ayres[7] review various correlations of wind tunnel data to flight drag correlations for unswept, swept and delta wings, boattail effects, supercritical wings, sting support and Reynolds effects. Lombardi[8] experimentally gives effects of blockages on forces and moments for AGARD calibration models of different sizes for subsonic and transonic cases. Duraisamy et al.[9] has studied the wall interference effect on subsonic unsteady airfoil flows using a Navier-Stokes solver for the flow field over the airfoil and comparison with experimental results are also reported. It is seen that most of the studies are limited to low speed only and not much information of blockage and tunnel wall interference effect on pressure distribution on the body is reported especially in supersonic speeds.

Model Design

A CAD of standard full scale model of 53 feet semi-trailer truck was designed for CFD study. The major dimensions of the model are shown in Figure 1. The complex curvatures of actual semi-trailers which are not necessary to observe flow structures were avoided in order to simplify the model and computational effort. Computational model was built by considering two different blockage ratios (1.875 % and 15 %). 1.875 % blockage ratio represents relatively practical case where as worst case scenario is demonstrated by 15 %

For each case the length of the computational domain is six times of the same of model. The ratio of width to height of wind tunnel is kept as 1:1. Figure 2 illustrates the cases of domain having 1.875 % and 15 % blockage ratios.

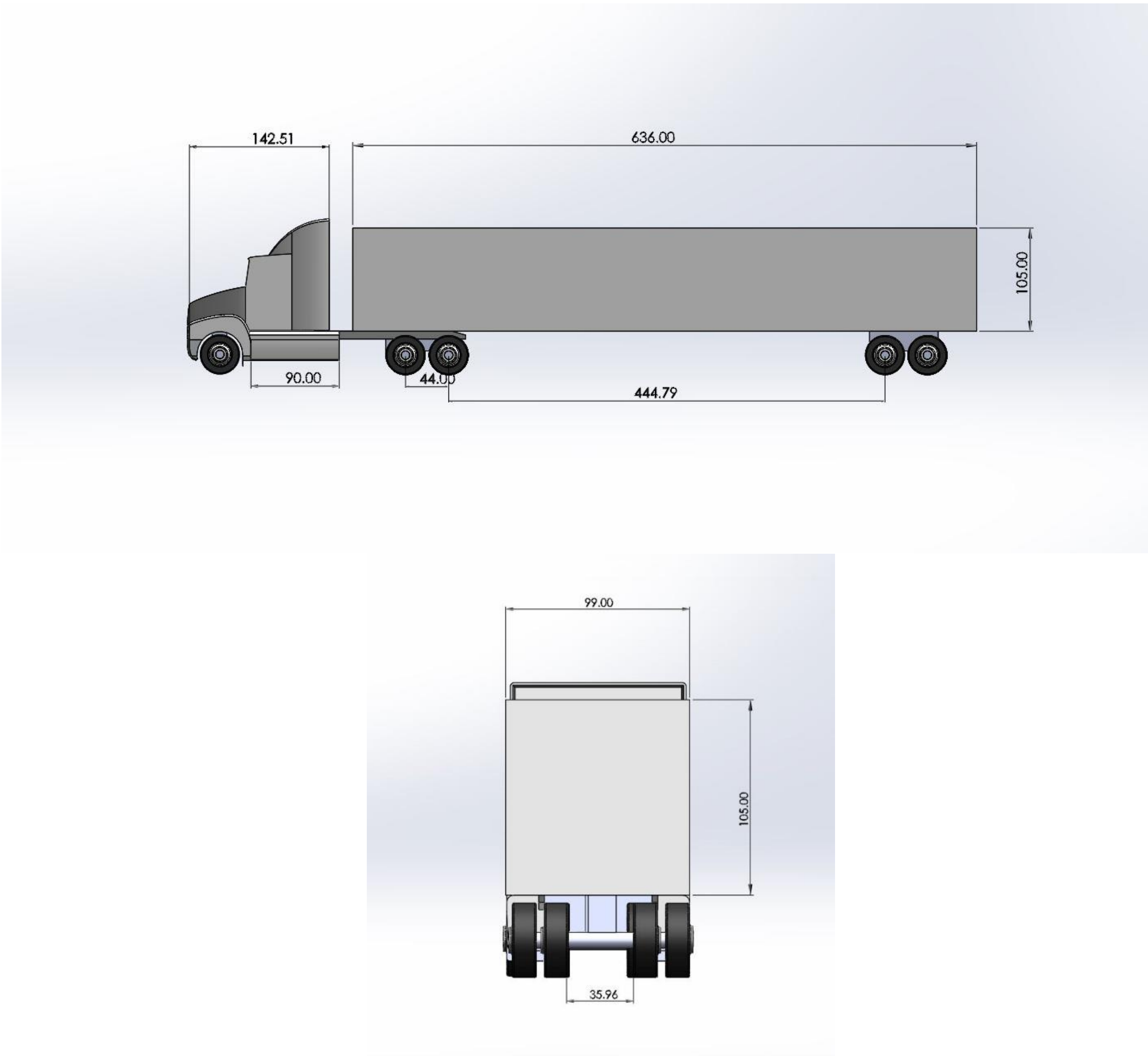
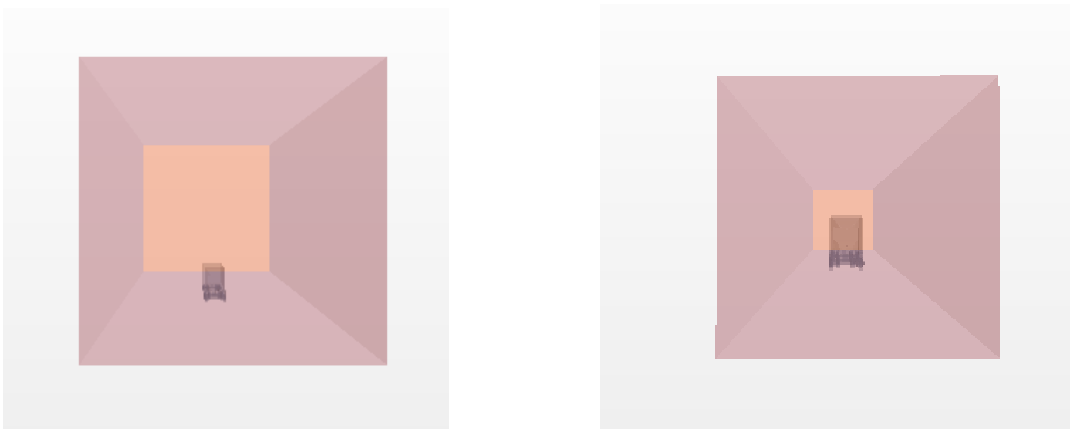


Figure 1: CAD model for Semi-trailer (Dimensions are in inch)



(a) (b)

Figure 2: Computational Model for different blockage ratios (a) 1.875 % (b) 15 %

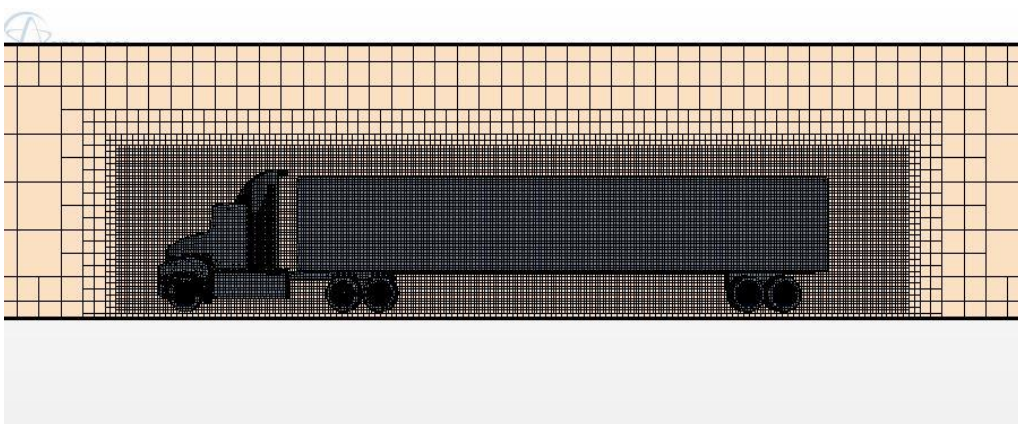
CFD Study

The computational models were simulated using STAR CCM+ for both 50 mph and 70 mph speed. Air is selected as the fluid for all cases. The following physical conditions were selected:

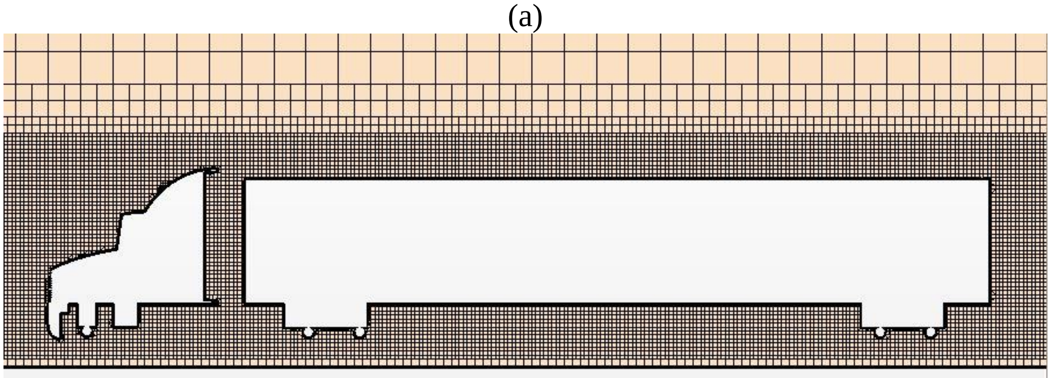
- Three dimensional
- Steady flow
- Turbulent flow
- Segregated flow

The two equation SST k-ω model was used for turbulence simulations. Reynolds-Average Navier-Stokes (RANS) equation was considered for flow solutions. The inlet of the wind tunnel was defined as velocity inlet. The surrounding surfaces of the domain was kept as wall with no slip condition. The ground was moving with same inlet velocity. The outlet of the domain was set as pressure outlet.

In terms of meshing, a mesh block was created surrounding the truck body. Surface Remesher, Prism Layer Mesher and Trimmer were considered as meshing model for creating around 2.7 million volume cells. Figure 3 depicts the volume meshing of the computational domain.



(a)



(b)

Figure 3: Volume meshing representation with truck model (a) Mesh Block (b) Prism layers

Simulation Results

The simulation was run for city speed (50 mph), high speed (70 mph) for each blockage ratio using the aforementioned physics and mesh settings. The following figures depict the pressure contours and velocity profile distribution for different cases.

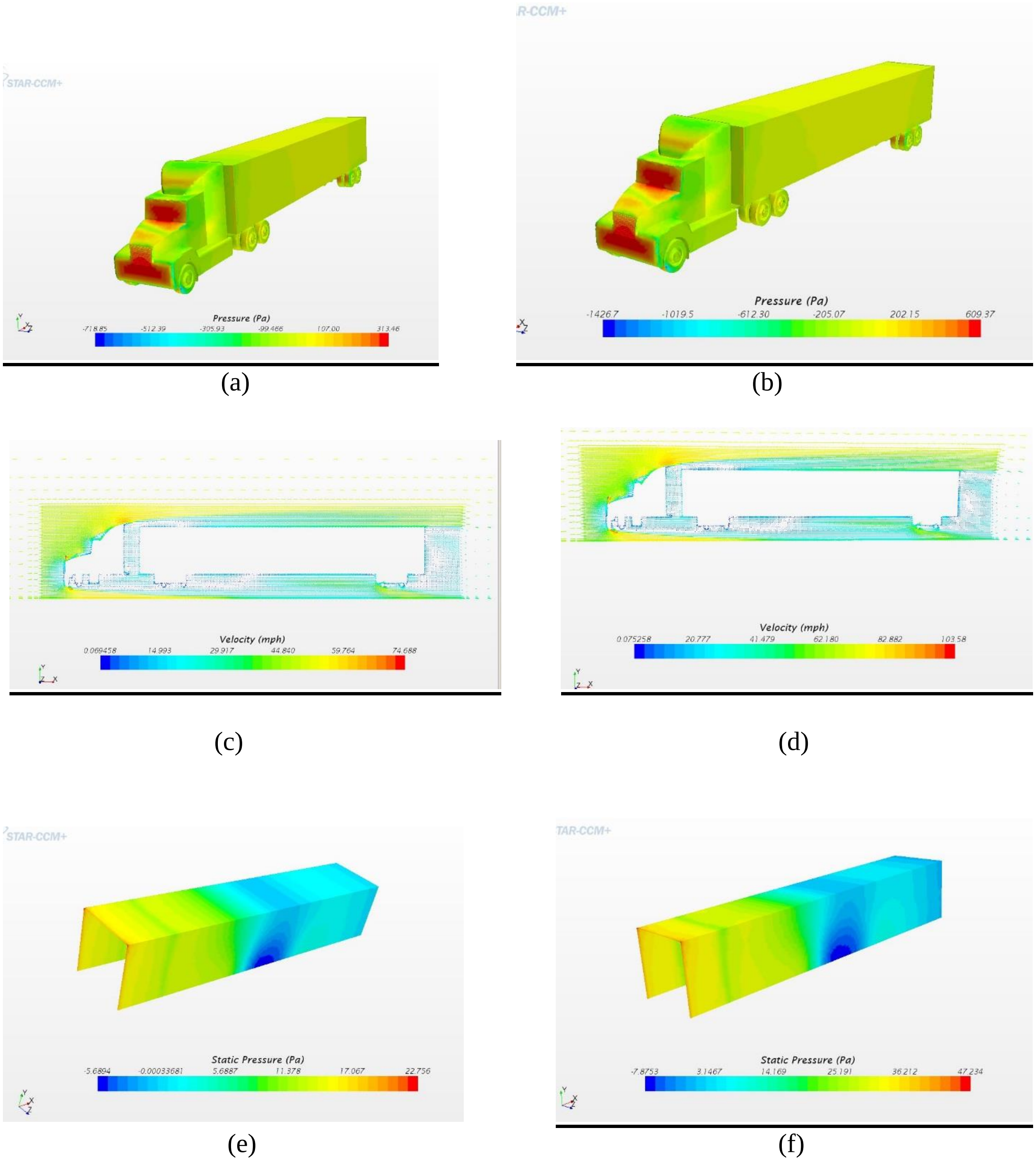


Figure 4: Pressure contours, Velocity distribution and Wall pressure signatures for 1.875 % blockage ratio. (a), (c), (e) for 50 mph (b), (d), (f) for 70 mph

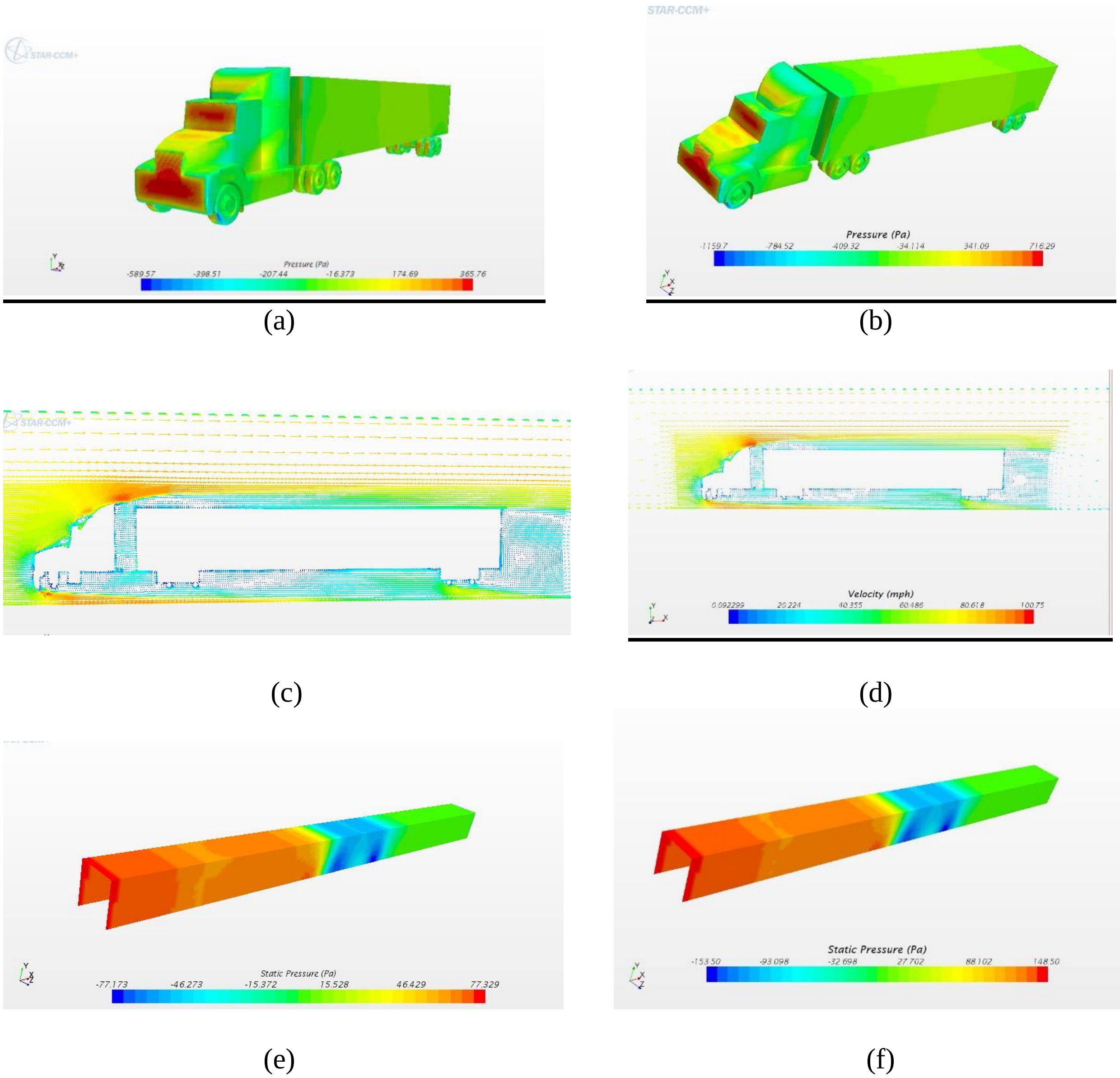


Figure 5: Pressure contours, Velocity distribution and Wall pressure signatures for 15 % blockage ratio. (a), (c), (e) for 50 mph (b), (d), (f) for 70 mph

The following table shows the drag co-efficient for different scenarios.

Table 01: Drag co-efficient (C_d) for different blockage ratio and speed

Case	50 mph	70 mph
15 % blockage ratio	0.85	0.84
1.875 % blockage ratio	0.60	0.58

From the pressure plots and velocity profiles for all cases, it was observed that there is a high pressure zone at that front part of semi-truck body which depict similar nature for all blockage ratios. For free domain case, the zone values were larger which make sense of the effect of having wind tunnel walls around the free flow field. For all cases, there is increase of velocity on the upside of the truck body causing the flow to move faster. Large flow separation regions are observed in the gap between the truck and trailer container, right behind the container and under body as well. With different blockage ratios, there is significant influences on wall pressure signatures. It can be noted that the pressure regions values are higher for 15 % blockage ratio.

It is no doubt that having different blockage ratios has significant effect on drag co-efficient. For high blockage ratio, the flow around the truck body is much affected resulting in an increase in drag co-efficient. The values of Table 01 shows the effect. Highest drag co-efficient was found for highest blockage ratio (15%).

Conclusion

Wind tunnel wall interference is crucial in determining actual flow structures and drag co-efficient for high blockage vehicles like semi-truck. This study shows the effect of different blockage ratios on flow structure and drag co-efficient. From the results, it can be stated the wall interference effect should be corrected in order to have right data in wind tunnel test. This correction can be done by wall-pressure signature method which can also be a future scope of this paper. The possible errors of the simulation might be generated from inadequate custom meshing, computational power limit etc.

References

1. Katz, Joseph and Walters, Robert "Effects of large blockage in wind-tunnel testing", Journal of Aircraft, Vol. 32, No. 5(1995), pp. 1149-1152, doi: 10.2514/3.46852

2. Hackett, J.E, "Recent developments in the calculation of low-speed solid-walled wind tunnel wall interference in tests on large models Part I: evaluation of three interference assessment methods", Journal of Progress in Aerospace Sciences Volume 39, Issues 6–7, August–October 2003, Pages 537–583, doi: 10.1016/j.paerosci.2003.08.001

3. Ewald, B. F. R., Ashill, Patrick Ralph, Binion, Travis, Cooper, Kevin Russell et al., "Wind Tunnel Wall Correction," AGARDograph 336, published by the North Atlantic Treaty Organization, October 1998.

4. Maciejewski M Osmolszi W and Poznanska P, "Numerical simulation of blockage effects in wind tunnels", Proc 14th European Simulation Symposium, Verbraeck A and Krug W eds. SCS Europe BVBA 2002.

5. Griffith M D, Houringan K and Thompson MC, "Numerically modeling blockage on flow between flat plates", 15 th Austrilian Fluid Mech Conf, U. Sidney, 13-17 Dec., 2004.

6. Amrouche N, Dizene R and Laneville A, "Observation of wind tunnel blockage effects on pressure distribution around rectangular prism in smooth and grid turbulent flows", Revue des Energies, Renouveleble SMEE'10 Bou Ismail Tipaza, pp21-26, 2010 1993.

7. Saltzman E J and Ayers T G, "A review of flight to wind tunnel drag correlations", AIAA Paper No 81-2475, doi: 1981-2475

8. Lombardi, Giovanni and Morelli, Mauro, "Analysis of some interference effects in a transonic wind tunnel", Journal of Aircraft, Vol. 32, No. 3 (1995), pp. 501-509, doi: 10.2514/3.46748

9. Duraisamy K, McCroskey W J Baeder, "Analysis of wind tunnel wall interference effects on subsonic, unsteady airfoil flows", Journal of Aircraft, Vol. 44, No. 5 (2007), pp. 1683-1690, doi: 10.2514/1.28143