OPEN ACCESS

International
Journal
Of Modern Engineering Research (IJMER)

Further Study of Wall Interference for High Blockage Vehicles in Closed Test Section Wind Tunnels

Wael Mokhtar[1], Chetan Madagi[2], Md Rakibul Hasan[3]
[1](PhD, Director of School of Engineering, Grand Valley State University, USA
[2,3] (MSE Student, Mechanical Engineering, Grand Valley State University, USA

ABSTRACT: *Wall interference is one of the vital factors that creates inaccuracy for wind tunnel measurement, which is a necessary tool to predict the aerodynamic performance in automotive and aircraft industries. Flow over high blockage vehicles such as buses and semi-trucks is complex and difficult to predict with accuracy. In order to have an efficient aerodynamic design, accurate data collection from wind tunnel is of a great importance. The current work performed a Computational Fluid Dynamics (CFD) study of a full scale generic semi-truck in order to assess the wall interference effect by analyzing the flow structures for several blockage ratios. The blockage ratios are generated by varying the areas of the wind tunnel cross section and keeping the model size same. A closed wall test section was used for the study. The simulation was run for highway speed for each blockage ratio. The commercial CFD software STAR CCM+ was used in the study.*

The analysis process in its entirety is detailed throughout this report, including methodology, procedures, and end results. A detailed description, including CAD model of generic semi-truck is provided as well. CFD study evolved throughout the process and the current findings are the result of intense engineering efforts. This report serves to document the entire process from initial background research to final recommendations for improvements.

Keywords: *Blockage ratio, Computational Fluid Dynamics (CFD), Wall interference, Wind tunnel.*

I. INTRODUCTION

In today's economic environment there is an intense need for highly fuel-efficient commercial vehicles. Every year, more than 2 million tractor-trailer trucks travelling on America's highways consume about 36 billion gallons of diesel fuel, representing more than 10 percent of the nation's entire petroleum use, according to studies by researchers at Lawrence Livermore National Laboratory[1]. The fuel consumption in heavy duty trucks depends mainly on aerodynamic drag, rolling resistances, power train losses, grade changes and accessory losses as per Cummins MPG guide[2] and out of these factors aerodynamic drag accounts for 70% of fuel burn when travelling at the highway speed of 70 miles per hour. Hence there is an immense need to reduce the drag and improve the fuel efficiency as for every 2% of reduction in drag help improve approximately 1% in fuel economy and this can be achieved by improving the aerodynamics of heavy duty trucks.

Wind tunnel is one of the main tools used in the aerodynamic research to study the effects of air moving around the test models and assess overall aerodynamic performance of test object. CFD and road testing are the other two tools currently used in these studies. Each has its limits and strengths. The integration between the three tools has been the way to get as much details, however, the cost is still a challenge.

Wall interference in wind tunnel testing is one of the major challenges being faced by the researchers as wind tunnel tests can be applied to any hypothesis but are limited by tunnel wall interference/blockage. The presence of test section walls changes the physical conditions that the test model experiences during tests and hence wall interference corrections need to be applied to test results whenever data is to be related to a free-air situation –or whenever a tunnel-to-tunnel data comparison is to be made[3] .

Over the years, several methods were introduced to assess the wall interference for both open and closed section wind tunnel. A good comprehensive summary of these methods was presented by Ewald et al in the AGARD special publications[4]. Another publications was published in the sixties for classical methods such as the method of images, Garner et al[5]. Britcher et al[6] studied the boundary corrections for wind tunnel testing of large ground vehicles in strong cross wind conditions. The focus on a common method that has been successfully used for closed wall test section, wall-signature method could be the future scope of the present study. It was originally introduced by Hackett in the late seventy [7, 8]. The method has been evolving and improved over the years. A recent model for it was automotive applications was presented recently by Cooper and Mokry[9].

In the wall pressure signature method the flow field about the test model is approximated using the superposition of the flows associated with the set of sources and sinks. The strengths and positions of these sources and sinks are determined so as to reconstruct the measured velocity distributions on the tunnel walls. Once determined the effect of the tunnel walls on the measured drag and dynamic pressure at the model is estimated and appropriate blockage corrections made[3].

In the present study, simulation on generic semi-truck model has been carried out to assess the wall interference effects by analyzing the flow structures and pressure distributions over the test model. The study mainly addresses flow structures and pressure signatures influenced due to wall interference.

Literature reviews, the CAD model of the semi-truck, simulation results along with the conclusion and recommendations for future improvement have been discussed in the following sections.

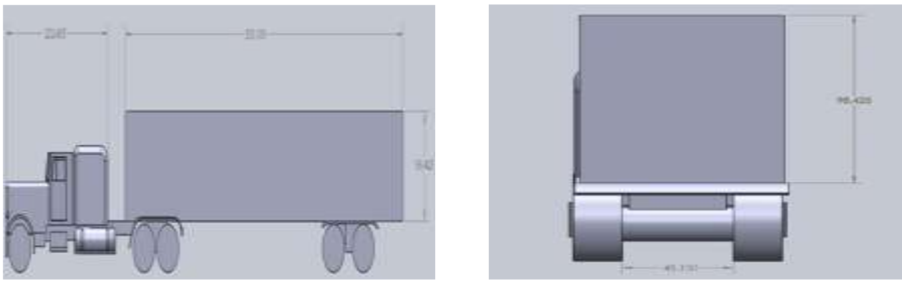
II. LITERATURE REVIEW

Adcock et al [3]. examined in detail the differences of fluid dynamics between free flight and wind tunnel tests. The study included the effects of area blockage in the tunnel, growth of model and tunnel wall boundary layers, and the effects of longitudinal slots in the test section wall. Kang et al developed the new blockage correction method for the wall interference correction of closed test-section subsonic wind tunnels based on the nonlinear relationship between separation blockage and the separation drag[10]. Allmaras revised and improved the two dimensional tests of bluff bodies[11]. The method used experimentally measured tunnel wall pressures to approximately reconstruct the flow field about the body with potential sources and sinks and with the help of these sources and sinks, the measured drag and tunnel dynamic pressure are corrected for blockage effects. Iyer et al applied WICS wall interference method to the National Transonic Facility and 14 by22 ft subsonic wind tunnel at the NASA Langley Research Center [12]. WICS wall interference method calculates free air corrections to the measured parameters and aerodynamic coefficients for full span and semi span models when the tunnels are in the solid wall configuration. Mokry devised a low order panel method for calculating wall interference corrections to the measured drag force in automotive wind tunnels with ¾ open or slotted wall test sections[13]. The calculations shown that drag measurements in closed test sections and ¾ open test sections require negative and positive corrections respectively and inside a test section with 30% open longitudinally slotted walls the measurement is nearly interference-free. Kong et al [14] tested the large truck-trailer combination model in a blockage tolerant wind tunnel. Results of the study indicates pressure distributions and drag coefficients are unaffected by blockage for values as high as 29%. Mokhtar et al [16] studied the effect of wall interference on the high blockage vehicles. Study performed a CFD simulation on a generic truck model and assessed the wall interference effect by analyzing the flow structure for two different blockage ratios. Other studies pertaining to wind tunnel wall interference and CFD study of ground vehicles helped in the development of the present study such as the simulation considerations for commercial vehicles in strong crosswind conditions by Britcher et al [17] and computational and experimental aspects of ground simulation for vehicles in strong cross wind conditions by Mau-Kuo Chen et al [18].

The above discussion is a sample of the effort taken to address wall interference in wind tunnel testing. The focus of the current work is to investigate the effects on the flow structures using CFD simulations of the wind tunnel environment. The details of the method and the studied cases are presented below.

III. SEMI-TRUCK MODEL

Fig 1 (a) and (b) shows details of the simplified model of a generic semi-truck developed for the investigation. This model represents geometric configuration without any aerodynamic drag reducing devices incorporated. Unnecessary curvatures and geometric parts that are not required for analyzing flow structures have been removed to simplify computational effort.

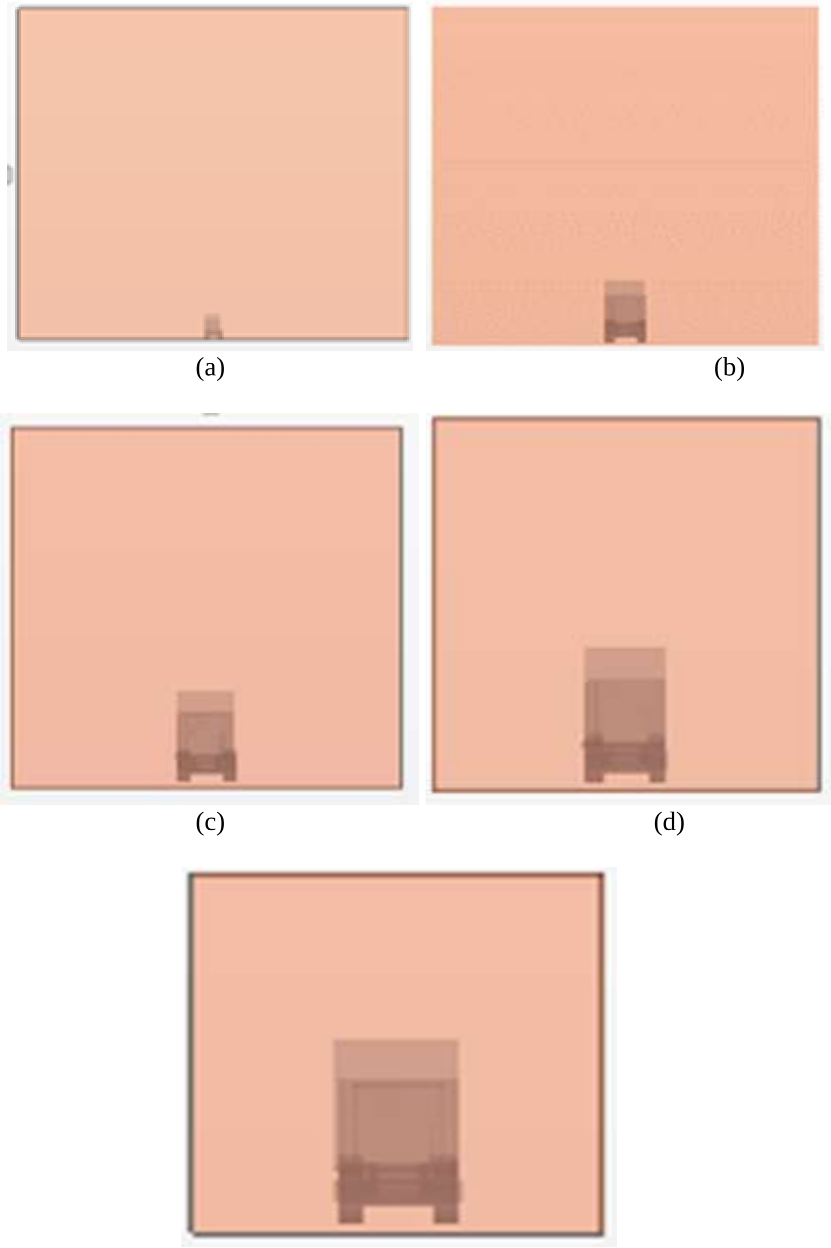


(a) (b)

Fig 1. Semi-trailer model major dimensions. (a) Side view (b) Rear view (Dimensions are in inch)

IV. WIND TUNNEL MODEL

A numerical closed-wall test section is developed to study a wide range of blockage ratios. The semi-truck model sizes was fixed in all the studies to maintain the same Reynolds number and the cross area of the test section was changes. The study included blockage ratios: 15%, 7.5%, 3.75%, 1.875% and 0.3%. In this study 15 % blockage is considered to be the worst case scenario and 1.875% representing nearly practical case. For every case the length of the computational domain is constant and is six times larger than the full length of semi-truck model. The ratio of width to height of wind tunnel is kept as 1:1. Fig 2 illustrates the cases of wind tunnel having 0.3%, 1.875%, 3.75%, 7.5%, and 15% blockage ratios respectively.



(a) (b)

(c) (d)

(e)

Fig 2. Computational model for different blockage ratios (a) 0.3% (b) 1.875% (c) 3.75% (d) 7.5% (e)15%

V. CFD MODELING

The numerical test sections were simulated for a highway speed of 70 mph using STAR CCM+ software package. The flow field of the semi -truck model simulation involves solving of the set of partial differential equations with pre-defined boundary conditions. Outer boundaries of the computational domain and

CFD mesh are generated for the external aerodynamic simulation. Mesh models selected for the study are Surface Remesher, Trimmer and Prism Layer Mesher was used to better resolve the flow near the walls. Fine meshing is done to capture very minute details and 19 million volume cells have been generated. Volume meshing of computational domain is as illustrated in fig 3.

The outer boundaries of the computational domain are considered to be the walls of a virtual wind tunnel. The inlet for the wind tunnel is defined as the velocity inlet and outlet is defined to be the pressure outlet. The surrounding surfaces of the domain were kept as wall with no slip condition and ground is considered to be moving as same inlet velocity to simulation a moving ground wind tunnel test section.

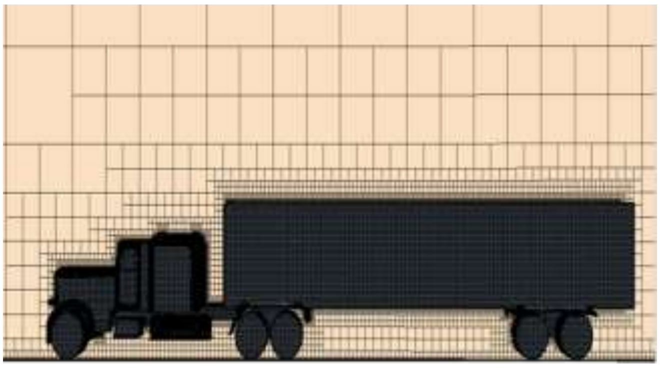


Fig 3. Representation of volume mesh with prism layers

5.1 Physics and Boundary Conditions.

Star Ccm+ uses the finite volume method for solving and is known for its computational power in simulating different fluid dynamics phenomena. In this study air is considered as the fluid for all the cases. Selected physical conditions are as follows:

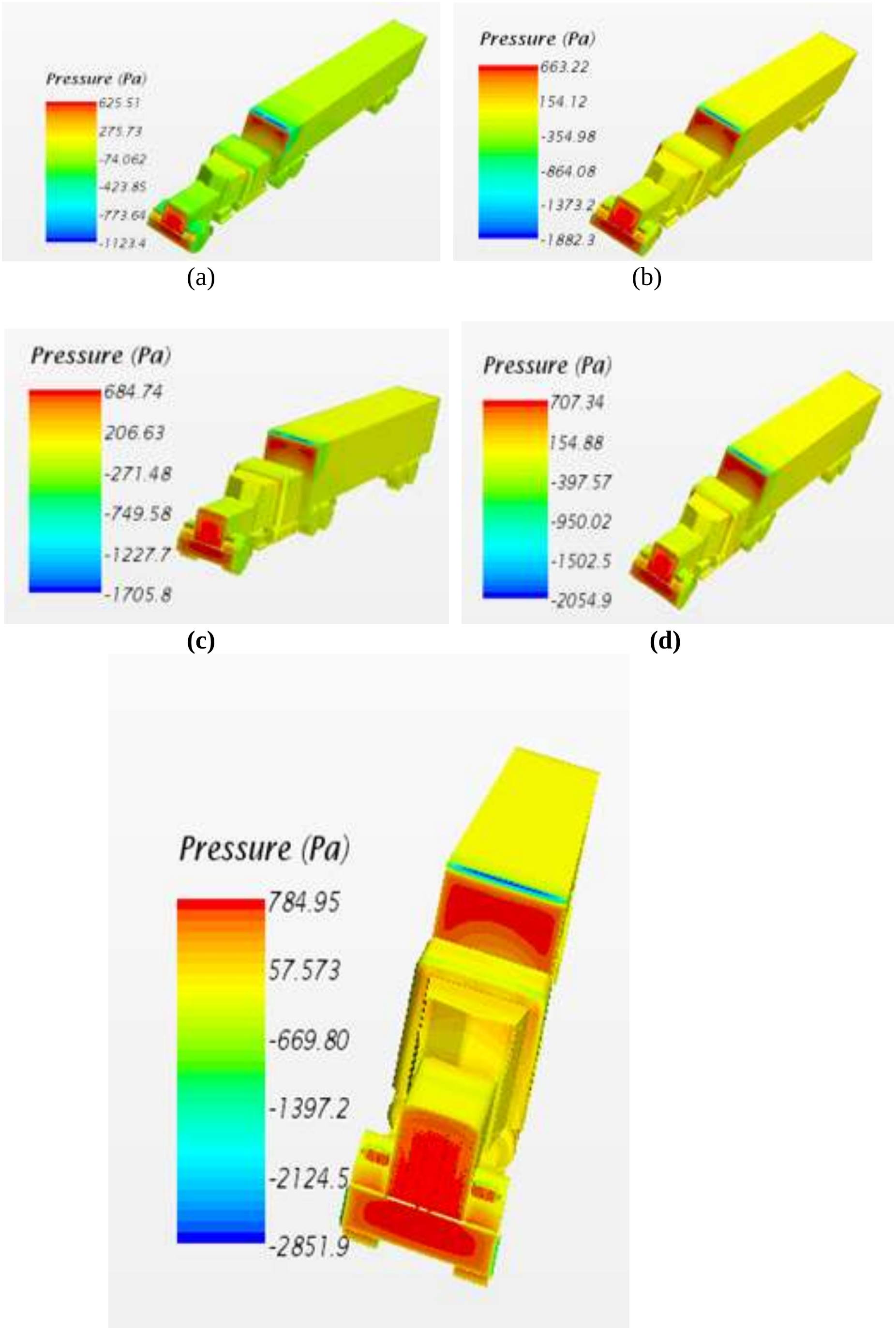
- Three Dimensional
- Steady flow
- Turbulent flow
- Segregated flow
- Two equation SST k-ω
- Reynolds-Average Navier Stokes (RANS)

VI. SIMULATION RESULTS

As it is a well-known fact that the vehicle manafacturers constantly strive to reduce the aerodynamic drag which directly influences the fuel economy, pressure and frictional resistance can be considered as the important factors in determination of aerodynamic drag. Pressure resistance is greatly influenced by the pressure distribution in the boundary layers of the semi-truck body. Fig 4 depicts the pressure distribution on the boundary layers of the semi-truck under study. The pressure distribution for all the cases are scaled later to see the actual pressure variations on the test model due to the wind tunnel blockage effects.

Fig 5 (a) depicts the close view of high pressure regions for 0.3% blockage ratio as indicated by arrows and when compared to high pressure regions of 1.875% blockage ratio as depicted in Fig 5(b), there is not much difference in the pressure but there is slight variation which can be clearly seen with the help of the Figs. The pressure distribution on the boundary layers is smooth and not much wall interference is observed and hence case with 0.3% blockage can be considered as near to blockage free condition when compared to case with 1.875% blockage.

6.1 Pressure contours for different blockage ratios.

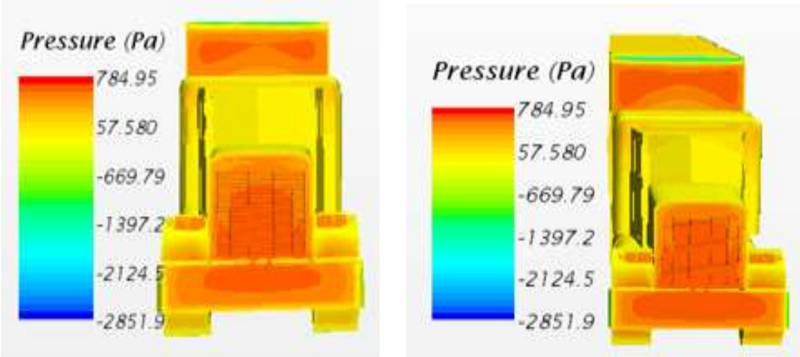


(a) (b)

(c) **(d)**

(e)

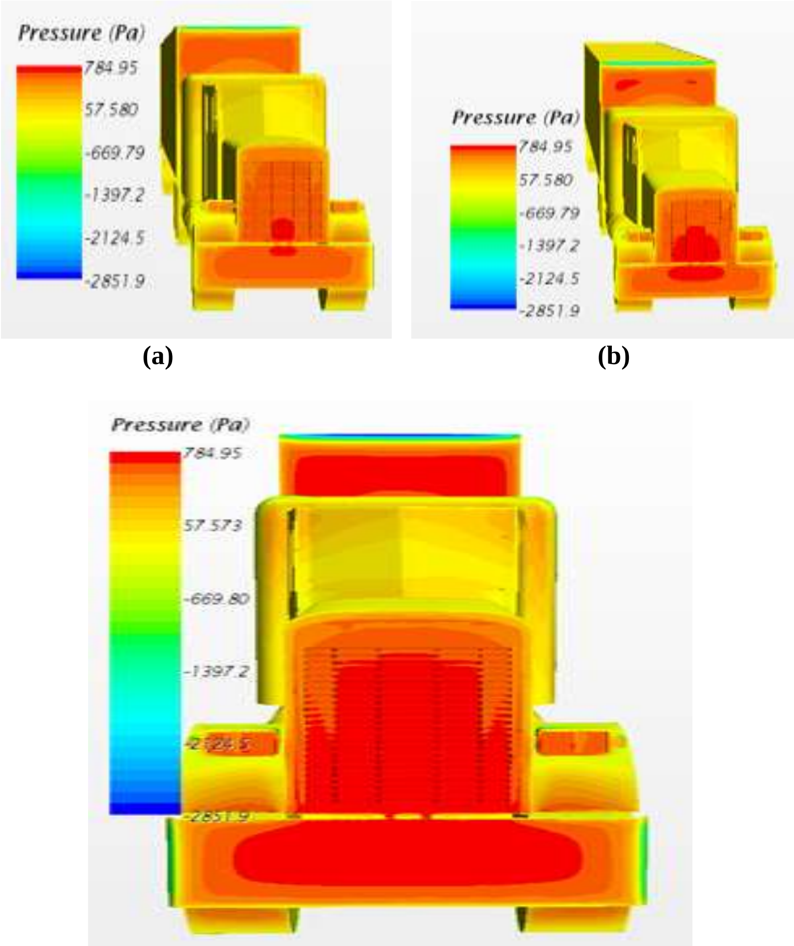
Fig 4. Pressure contours for different blockage ratios(a) 0.3% (b) 1.875% (c) 3.75% (d) 7.5% (e)15%



(a) **(b)**

Fig 5 High pressure regions (Pressure Scaled)

When looked at 3.5%,7.5% and 15% blockage ratios effect on high pressure regions as depicted in the Fig 6 (a),(b) and (c), it can be easily inferred from the Figs that the blockage effects has caused increase of pressure on respective models and 15% blockage ratio causing high pressure on the model compared to all lesser blockage models. The sudden rise in the pressure in the high pressure regions are due to shocks from tunnel wall hitting the model geometry. 0.3% blockage model is least effected as it is almost considered to be wall interference free case.

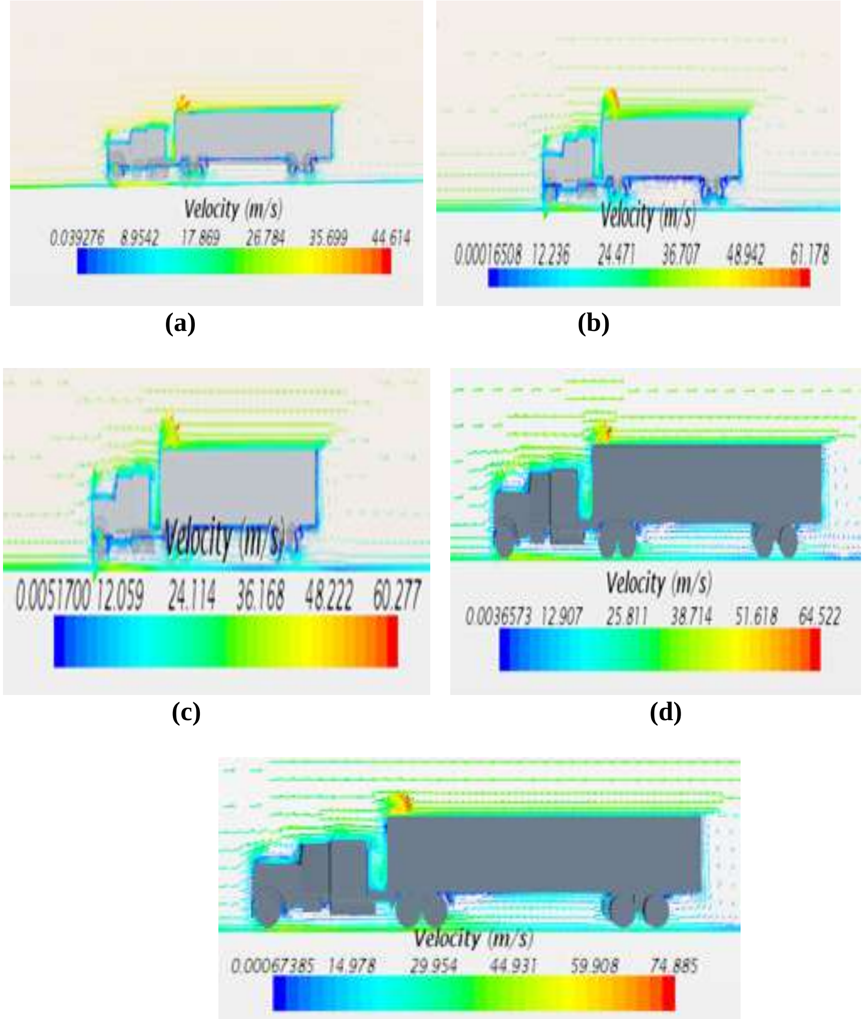


(a) **(b)**

(c)

Fig 6 closed view of high pressure regions (Pressure Scaled)

6.2 Velocity Vectors



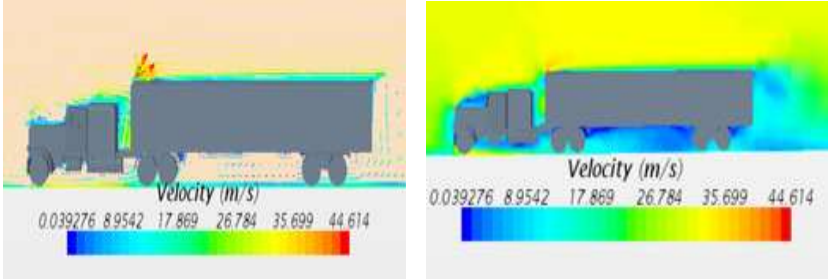
(a) **(b)**

(c) **(d)**

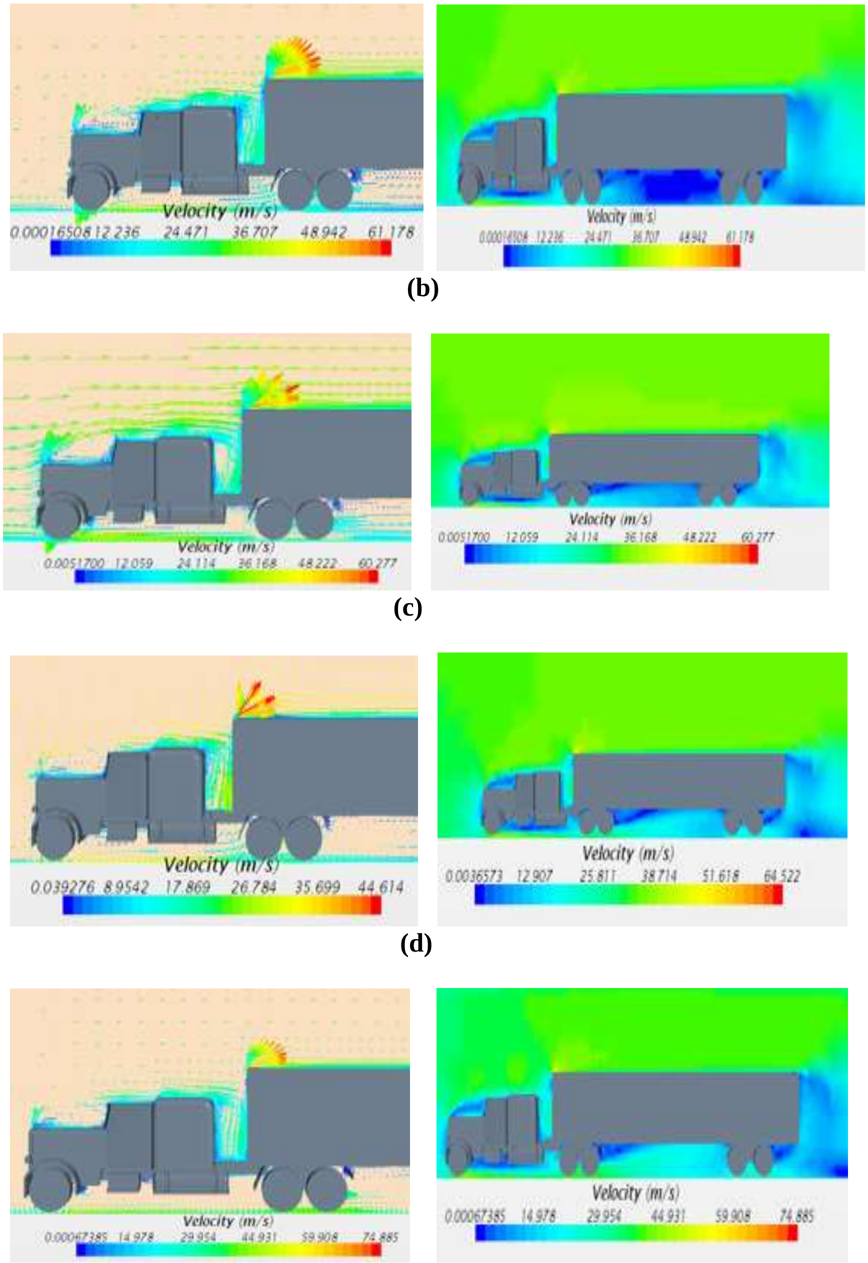
(e)

Fig 7 velocity vectors

When observed the flow at the regions denoted by arrows in Fig 7 closely with the help of velocity vectors and with integral convolution, it can be inferred that the flow around the frontal area of the vehicle having 0.3% blockage effect is relatively steadier than the other cases. In front of the truck there is a low velocity flow can be seen which is the result of high pressure stagnation. Flow separations observed are caused due to sharp corners of the semi-truck and stream of vorticity trailing every corner in the model. Long vorticity trail in the separation regions and behind the wheel cover can also be seen. Flow is affected by the gap between the truck and the trailer and hence it gives rise for higher drag, flow is more effected for the case of 15% blockage giving rise to higher drag compared to rest of the cases and 0.3% being the least effected. The 15% blockage has caused confinement of flow around the model by reducing the area for the air to flow compared to 0.3% blockage ratio and hence, by continuity and Bernoulli's equation increases the velocity of the flow around the test model. Fig 8 depicts close view of region of interest.



(a)



(b)

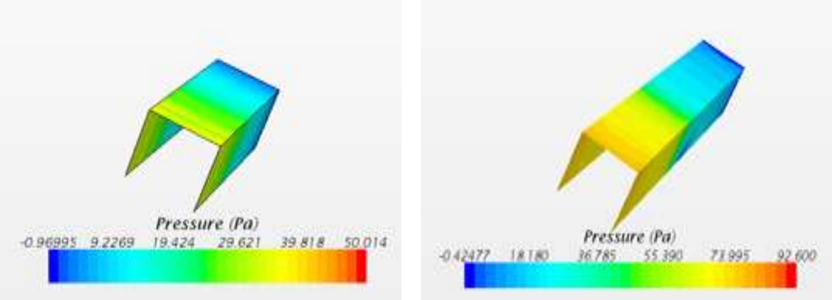
(c)

(d)

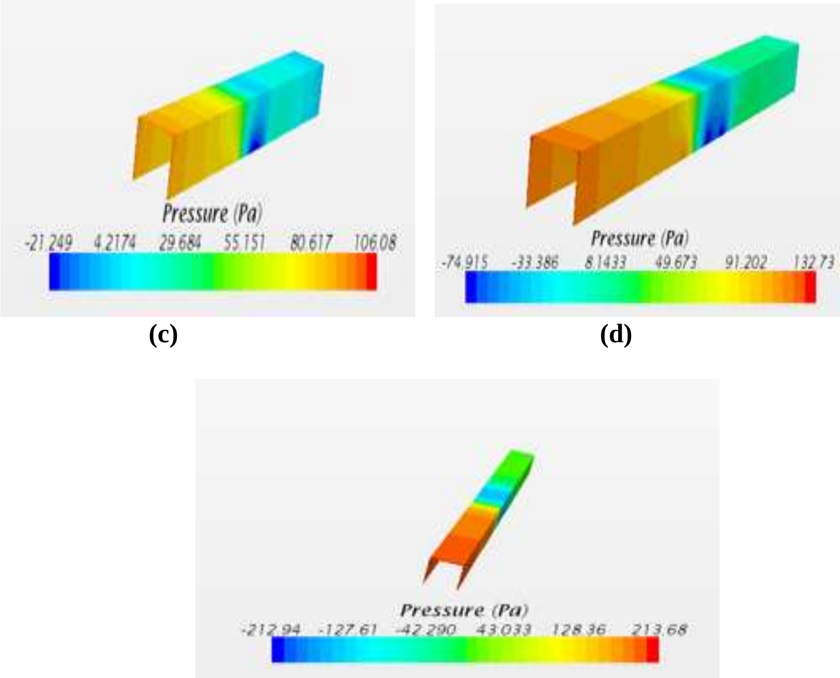
(e)

Fig 8. Closed view of velocity vectors (a) 0.3% (b) 1.875 (c) 3.5% (d) 7.5% (e) 15%

6.3 Wall pressure signatures



(a) **(b)**

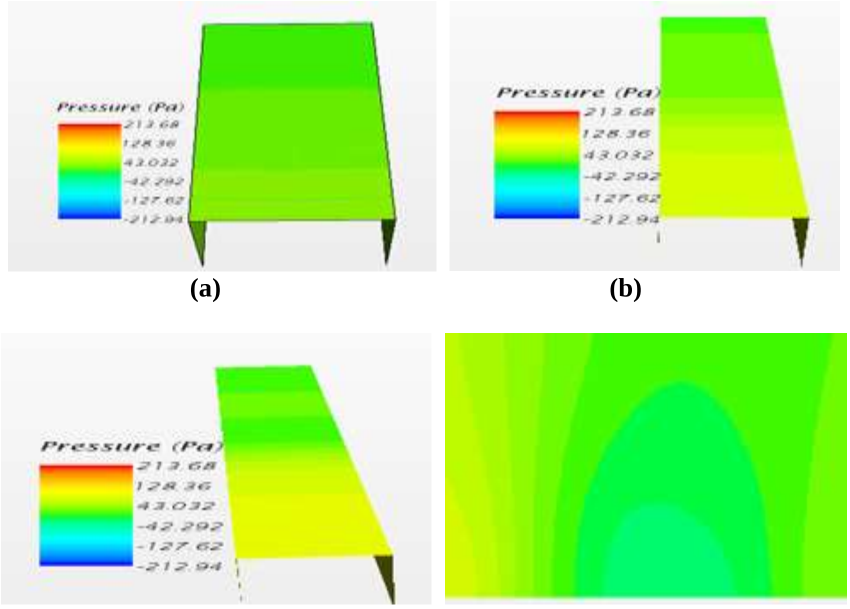


(c) (d)

(e)

Fig 9 pressure on the wind tunnel walls (a) 0.3% (b) 1.875 (c) 3.5% (d) 7.5% (e) 15%

By observing keenly at the higher pressure variations for all the cases as denoted by the arrows in the Fig 9 by scaling the pressure, pressure signatures on the wall of 0.3% blockage is compared with pressure signatures on the walls of 1.875% blockage and 3.75% blockage as depicted in the Fig 10 (a) and 10 (b) and 10 (c) it can be inferred that the increase in pressure in front of the blockage in case of 3.75% blockage ratio is mainly due to stagnation of flow at the front of blockage. This stagnation of flow causes high pressure in its vicinity and decreases eventually as it moves away from the high stagnation point. Gradual decrease in pressure can be observed which is due to sudden changes in the flow and increase of velocity. Effect of blockage ratios on the pressure signatures of the walls of wind tunnel can be clearly seen in the Fig 10 (c) and not drastic difference in pressure signatures of 0.3% and 1.875% blockage ratios could be deduced but there is a slight variations due to blockage effects causing sudden variations of flow.

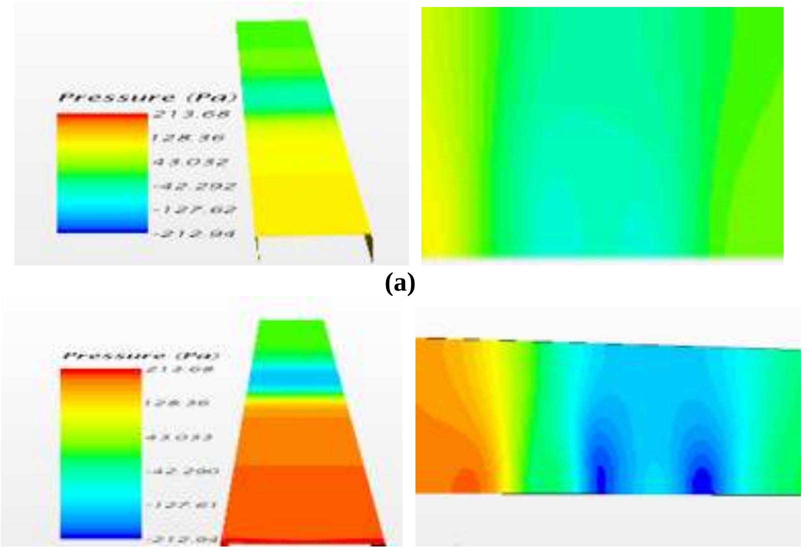


(a) (b)

(c)

Fig 10. Closed view of pressure signature on walls

Now comparing the blockage effects of 7.5% blockage ratio and 15% blockage ratio on the wall pressure signatures as depicted in the Fig 11 (a) and (b), it can be inferred that the high pressure due to flow stagnation in the 15% blockage ratio is considerably higher than all the cases. Steep pressure decrease after approximately half the length of the wall is due to sudden changes in flow and increase in velocity at the edges. At the near end of the wall pressure stabilization can be observed and it is due to the fact that pressure was not significantly affected by change in flow.

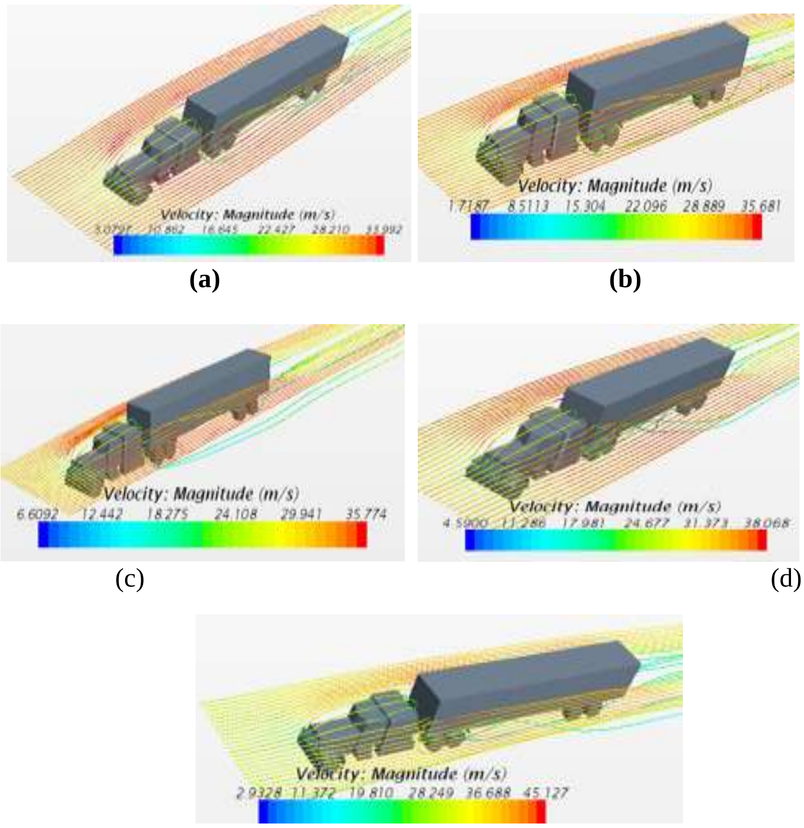


(a)

(b)

Fig 11. Closed view of pressure signature on walls

6.4 Streamlines on the semi-truck.



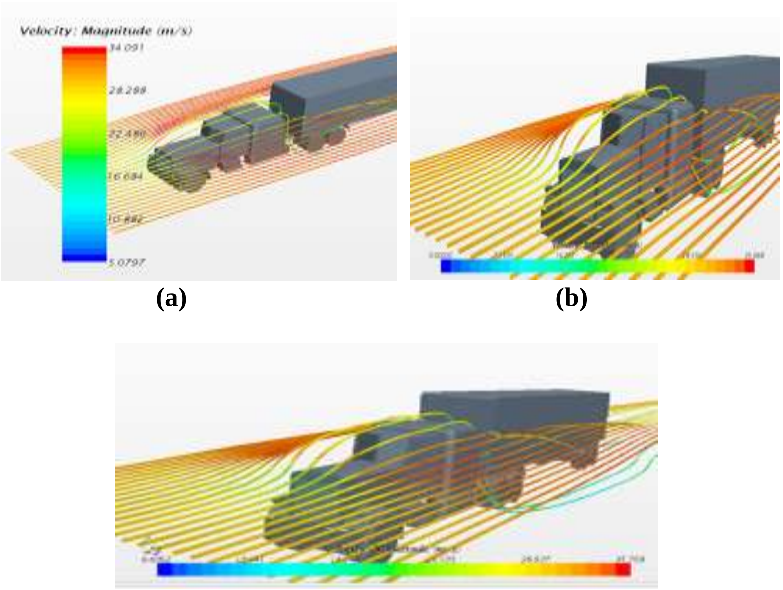
(a) **(b)**

(c) (d)

(e)

Fig 12 streamlines (a) 0.3% (b) 1.875 (c) 3.5% (d) 7.5% (e) 15%

When the effect of wall interference on the streamlines was observed more closely, it can be inferred that the there is not much difference in the streamlines for the cases with 0.3%, 1.875% and 3.75% blockage ratios but there is velocity increase in each respective case due to blockage and the regions of flow separation can be clearly seen as depicted in the Fig 13 (a),(b) and (c).

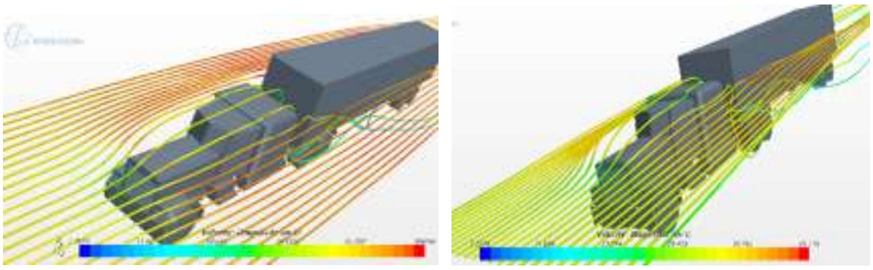


(a) **(b)**

(c)

Fig 13. Closer view of streamlines

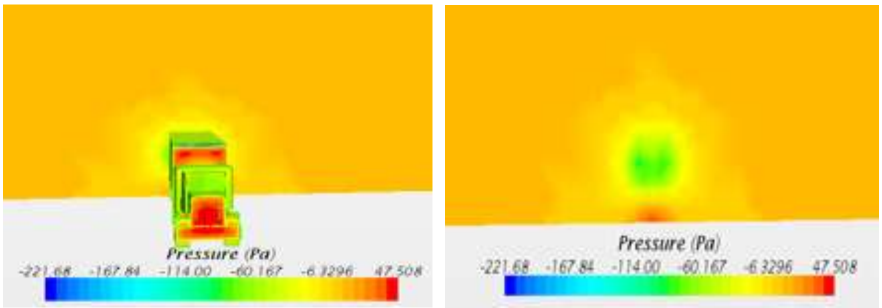
Now by comparing the 7.5% blockage ratio and 15% blockage ratio effect on streamlines and by viewing very closely, it can be inferred that the flow separation at the truck front is more significant in the case of 15% blockage ratio and very high velocity. The flow on the around the trailer remains same and attached at fairly large portion. Close view of 7.5% blockage ratio and 15% blockage ratio can be seen in the Fig 14 (a) and (b).



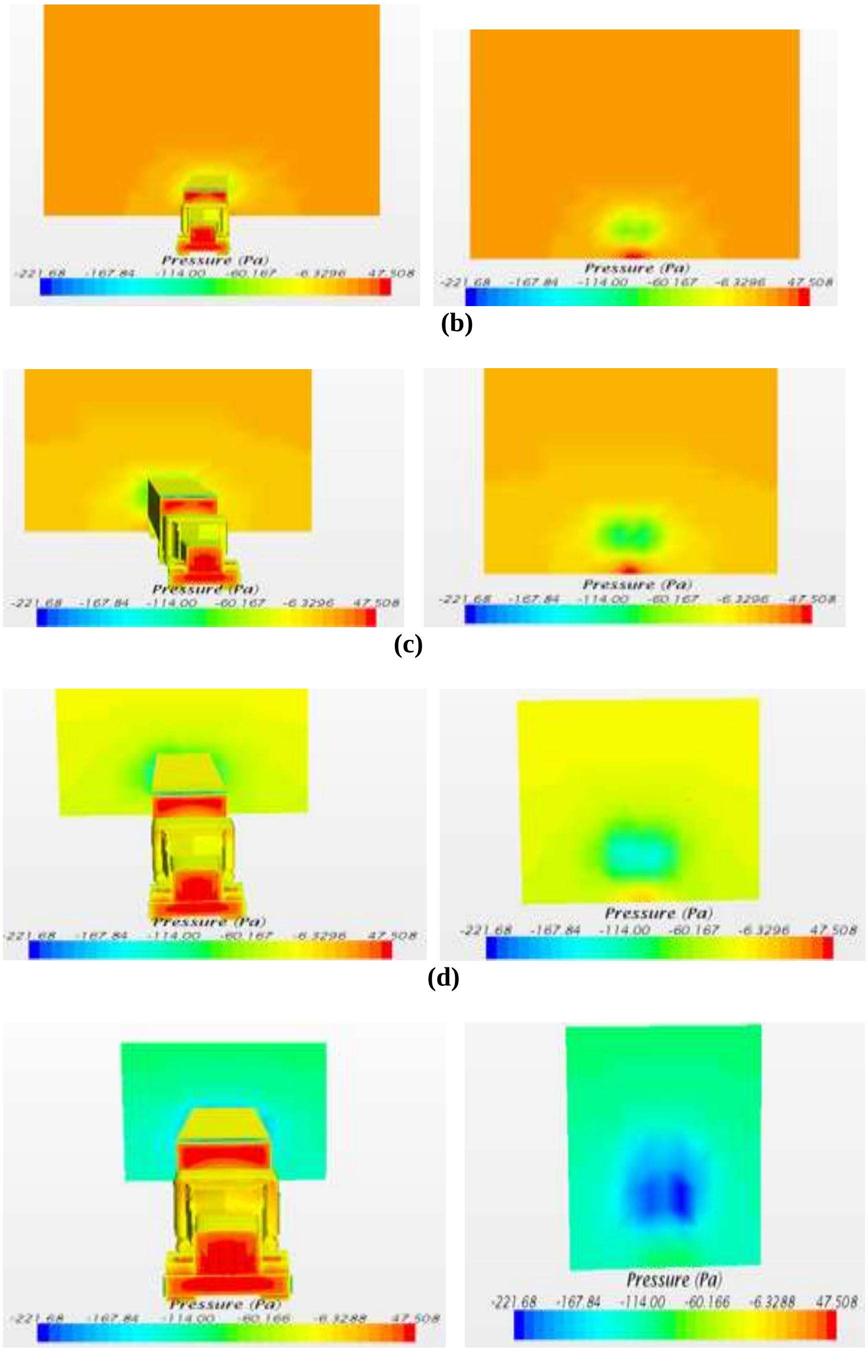
(a) **(b)**

Fig 14. Closer view of streamlines

6.5 Wake pressure



(a)



(b)

(c)

(d)

(e)

Fig 15. Wake pressure on the wind tunnel walls (a) 0.3% (b) 1.875 (c) 3.5% (d) 7.5% (e) 15%

The flow separation can be seen very clearly at the rear of the vehicle for different cases when wake pressure for all the cases is scaled. The separation is highly influenced by the pressure distribution imposed by the outer layer of the flow. The turbulent boundary layer withstands much higher pressure without separation compared to laminar flow. This phenomenon of separation causes the flow to change its behavior behind the vehicle affecting the flow field around the vehicle. This phenomenon is the important factor in studying the wake of the vehicle[15]. Observing the wall interference effect on wake pressure, it can be seen from the Fig 15 for all different cases that there is a high pressure stagnation right at the bottom for 0.3%, 1.875% and 3.5% blockage ratios and less pressure region on top. The high pressure stagnation would be the result of high accelerating flow from truck underbody hitting the wake plane causing increased base pressure and hence less base drag. The low pressure region is because of high stream velocity caused due to wall blockage, by Bernoulli's principle, and this lowered pressure arising as the boundary layer later becomes wake and grows on the model. Observing keenly at the last two cases having 7.5% and 15% blockage ratios, the high wall interference on the flow field has suppressed the wake structure and there is a negative pressure region in the wake because of which it tries to pull the vehicle back causing higher drag.

The distortion of wake can be seen in all the cases when compared to 0.3% blockage ratio indicating the effect of wall interference in the closed test section as for the ideal wind tunnel the wake is freely formed and not affected by walls.

VII. CONCLUSION/FUTURE IMPROVEMENT

Study concludes that the wind tunnel walls have significant effect on the aerodynamic drag and flow structures. Study shown the effect of different blockage ratios on the flow structure, pressure distribution on the test model along with the pressure signatures on the wind tunnel walls. The results clearly shows the variance of flow pattern, pressure distribution due to different blockage ratios and hence it can be concluded that the wind tunnel wall interference have adverse effect on drag coefficients and needs to be corrected.

Future scope of the study would be considering the methods of blockage correction and comparing with present results for higher accuracy of the test data.

REFERENCES

[1]. American Physical Society, “How to Save Billions of Gallons of Gasoline” https://www.aps.org/units/dfd/pressroom/news/2014/upload/2pr-SaveGas2014.pdf, accessed May 2016..

[2]. Cummins MPG Guide, "Secrets of Better Fuel Economy", http://cumminsengines.com/uploads/docs/Secrets%20of%20Better%20Fuel%20Economy_whitepaper.pdf, accessed May 2016.

[3]. Adcock J.B. and Barnwell R.W. 1984, Effects of boundary layers on solid walls in three dimensional wind tunnels, AIAA Journal Vol 22, No 3.

[4]. Ewald, B. F. R., et al., "Wind Tunnel Wall Correction," AGARDograph 336, published by the North Atlantic Treaty Organization, October 1998.

[5]. Garner, H.C., Rogers, E.W., Acum, W.E.A., and Maskell, E.C., "Subsonic Wind Tunnel Wall Correction," AGARDograph 109, published by the North Atlantic Treaty Organization, October 1966

[6]. Colin P. Britcher and Wael A. Mokhtar, Boundary Corrections for Wind Tunnel Testing of Large Ground Vehicles in Strong Crosswind Conditions, AIAA Science and Technology Forum 2015, AIAA no 2024128, Kissimmee, FL, Jan 5 to 9, 2015.

[7]. Hackett, J. E., Wilsden, D. J., and Lilley, D. E., “Estimation of Tunnel Blockage from Wall Pressure Signatures: a Review and Data Correlation, “ NASA CR-152241, Lockheed-Georgia Company, Marietta, Georgia, March 1979.

[8]. Hackett, J. E., Wilsden, D. J., and Stevens, W. A., "A Review of the Wall Pressure Signature and other Tunnel Constraint Correction Methods for High Angle-of-Attack Tests," AGARD-R-692, Munich, May 1980

[9]. Cooper, K. and Mokry, M., An Influence Matrix Method for Determining Boundary Interference on Automotive Models in Solid-Wall Wind Tunnels, SAE Technical Paper 2014-01-0611, 2014

[10]. Kang, Seung-Hee; Kwon, oh-Joon and Ahn, Seung-Ki, Blockage Correction Method for Separated Flows over an Aircraft in a Closed Test-Section Wind Tunnel, Journal of the Korean Society for Aeronautical and Space sciences, Volume 33, Issue 8, 2005, pp.42-49

[11]. Allmaras, S. ,"On Blockage Corrections for Two Dimensional Wind Tunnel Tests Using the Wall-Pressure Signature Method", NASA Technical Memorandum 86759, March 1986.

[12]. Venkit Iyer , Lockheed Martin, Joel L. Everhart, Application of Pressure-Based Wall Correction Methods to two NASA LANGLEY Wind Tunnels, 19th AIAA Applied Aerodynamics Conference,11-14 June 2014.

[13]. M. Mokry, Wall interference correction to drag measurements in automotive wind tunnels, Journal of Wind Engineering and Industrial Aerodynamics, 56(1995) 107-122.

[14]. Lingzhe Kong and Geoffrey V. Parkinson, Road vehicle model testing in a blockage tolerant wind tunnel, Proceedings of the First IAWE European and African Regional Conference, Volume 57, Issues 2–3, July 1995, Pages 137–143.

[15]. Pramod Nari Krishnani., Cfd Study of Drag Reduction of a Generic Sport Utility Vehicle, Ph.D. thesis, Mechanical Engineering Department, Mumbai University, Mumbai, India 2006

[16]. Wael A. Mokhtar and Md Hasan, A CFD Study of Wind Tunnel Wall Interference, ASEE North Central Conference, Paper no: 16, Mt Pleasant, MI, March, 2016.

[17]. Colin P. Britcher and Wael A. Mokhtar, Simulation Considerations for Commercial Vehicles in Strong Crosswind Conditions, SAE Commercial Vehicle Engineering Congress, SAE paper no 2014-01-2452, , Rosemont, IL, October 2014.

[18]. Mau-Kuo Chen, Wael A. Mokhtar, Colin P. Britcher, and Joe Mc Garry, Experimental and Computational Aspects of Ground Simulation for Vehicles in Strong Crosswind Conditions, SAE World Congress, SAE paper no 2014-01-0588, , Detroit, MI, April 2014.