Aerodynamic Analysis of Drag Reduction Devices on the Simplified Body for Tractor and Trailer by Using CFD

Hamidreza Shiri

Submitted to the
Institute of Graduate Studies and Research
in partial fulfillment of the requirements for the degree of

Master of Science
in
Mechanical Engineering

Eastern Mediterranean University
December 2016
Gazimağusa, North Cyprus

Approval of the Institute of Graduate Studies and Research

Prof. Dr. Mustafa Tümer
Director

I certify that this thesis satisfies the requirements as a thesis for the degree of Master of Science in Mechanical Engineering.

Assoc. Prof. Dr. Hasan Hacışevki
Chair, Department of Mechanical Engineering

We certify that we have read this thesis and that in our opinion it is fully adequate in scope and quality as a thesis for the degree of Master of Science in Mechanical Engineering.

Assoc. Prof. Dr. Hasan Hacışevki
Supervisor

Examining Committee

1. Prof. Dr. Fuat Egelioğlu
2. Assoc. Prof. Dr. Hasan Hacışevki
3. Asst. Prof. Dr. Neriman Özada

ABSTRACT

Nowadays, the demand of using fossil fuels has been increased all around the world. The most important issue in the automotive industry is the reduction of fuel consumption of vehicles. Because of this reason, development of more fuel efficient vehicles has been emerged.

Most of the company's focus on the aerodynamic analysis and modifications of truck body, while trailer aerodynamics has been pushed a side. In this study further research on the trailer aerodynamics modifications for drag reduction is conducted.

Different aerodynamic truck modifications and trailer devices such as Deflector, Gap Fairing, Vortex Stabilizer, Frame Extension, and Teardrop have been employed. Computational fluid dynamics theories have been used to investigate the effects of the aforementioned modifications on flow structures around the truck and trailer. In this study speed of 90 km/h, used with 0^o yaw angle for the simplified truck and trailer to compute the aerodynamic drag for each combination of the devices to determine the optimum drag coefficient.

The results revealed that the combination of the deflector and the Frame Extension have shown the maximum drag reduction. The gap between truck-trailer and the rear of the trailer are the most susceptible area to reduce aerodynamic drag. Therefore, these two devices have been implemented to understand the effect of these two regions.

Furthermore, in this study the significance of employing the truck modifications and trailer devices for aerodynamics improvement have been presented. Reducing gap area between truck-trailer, and base treatment of trailer has been employed to reduce drag, for instance. In this Study key findings are reduction of drag up to 39% with top fillet edge on truck and 42% reduction in aerodynamic drag with use of Deflector for truck and Frame extension for trailer. It is suggested for the future works to reduce the aerodynamic drag with additional simulation to compare the other combination of devices together.

Keywords: ANSYS-CFX®, Trailer devices, Aerodynamics, CFD, Truck, drag reduction.

ÖZ

Son zamanlarda tüm dünyada fosil yakıt kullanımı artmıştır. Otomotive endüstrisinin en önemli çalışmalarından biri de araçların yakıt tüketimini azaltmaktır. Bu nedenden dolayı yakıt verimli araçların geliştirilmesi çalışmaları başlamıştır.

Birçok üretici traktörün aerodinamiği ve gövdenin modifikasyonu üzerine yoğunlaşmış dorsenin aerodinamiği ile ilgili çalışmaları bir tarafa bırakmıştır. Bu çalışmada direnç azaltılması ile ilgili dorse aerodinamiği üzerinde araştırmalar yapılmıştır.

Dağıtıcı, ara kaporta, vortex sabitleyici, gövde uzatılması ve üst akım düzenleyici gibi değişik çekici modifikasyonları ve dorse elemanları üzerine çalışılmıştır. Çekici ve dorse etrafındaki söz konusu akış elemanlarının üzerindeki etkileri görebilmek için bilgisayar destekli akışkanlar dinamiği kullanılmıştır. Bu çalışmada araç hızı 90 km/sa ve sapma açısı da 0^0 olarak kullanılarak basit çekici ve dorse kombinasyonuna her bir durum için aerodinamik sürükleme dirençleri hesaplanarak optimum sürükleme katsayısı belirlenmiştir.

Neticeler ortaya çıkarmıştır ki dağıtıcı ve gövde uzatılması kombinasyonu en büyük sürükleme direncinin azalmasına sebep olmuştur. Çekici ve dorse arasındaki boşluk ile dorse arkasının aerodinamik sürklenmeyi azaltmaya en çok meyilli olan bölgeler olduğu görülmüştür. Dolayısı ile bu iki elemanın bu bölgelere etkili olduğu uygulamalarda görülmüştür.

Bu çalışmada çekici modifikasyonu ile dorse elemanlarının aerodinamiği iyileştirmeye etkisinin önemi vurgulanmıştır. Çekici ile dorse arasındaki boşluğun azalması ile dorse alt düzenlemelerinin sürükleme direncinin azalmasına etkileri gözlemlenmiştir. Bu çalışmadaki en önemli tesbit sürüklenme direncinin üst köşe dolgulu çekici için %39, dağıtıcı plakalı uzatılmış gövde için %42 azaldığı yönünde olmuştur. Daha sonraki çalışmalarda aerodinamik sürüklenme direncinin azaltılması için değişik düzenlemelerle simülasyon yapılması tavsiye edilmektedir.

Anahtar kelimeler: ANSYS-CFX®, Dorse, Aerodinamik, CFD, Çekici, Sürükleme direnci.

To my Family

ACKNOWLEDGMENT

I would like to express my gratitude to Assoc. Prof. Dr. Hasan Hacisevki Chair of Mechanical Engineering Department for his supervision, and guidance from the very early stage of this thesis to make this work completed. Above all and the most needed, he provided me constant encouragement and support in various ways. I am indebted to him more than he knows.

My parents, Mami and Dadi, receive my deepest love for their dedication and many years of unconditional support. You have been a source of inspiration throughout the years for me, and you show me the path toward success. I am indebted to you forever and I am proud of you in the whole entire life.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

NOMENCLATURES

V	Velocity of the Air
A	Frontal Area
L	Length of Truck and trailer
P	Pressure
H	Height of reference model
W	Width of reference model
k-ε	Turbulence Model
F_d	Drag Force
C_d	Drag Coefficient
Re	Reynolds Number
DP	Dynamic Pressure
CFD	Computational Fluid Dynamic

Chapter 1

INTRODUCTION

1.1 History of Vehicle Aerodynamics

Between 1900-1920 decades the productions of vehicles have been started. At the beginning their designs were like ships and plane. Initially, engineers were ignored the ground effect and the symmetry of the flow. Therefore, this incuriosity made some effects to increase the drag. Also, they ignored that when the wheels are out of vehicles chassis flow will distorted.

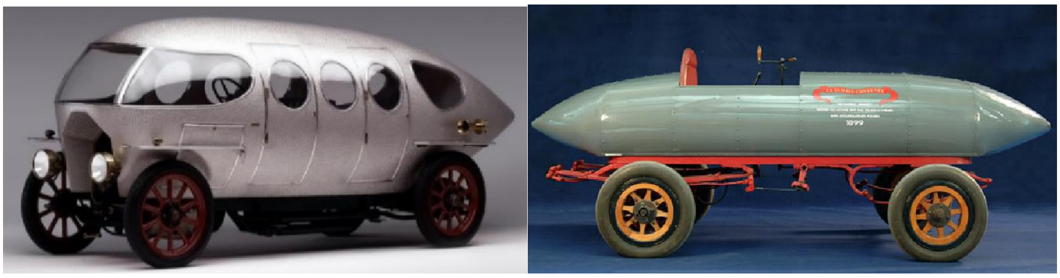


Figure 1.1. First prototypes of cars based in planes and ships forms (JamaisContente and Alfa Romeo of 1914) (taken from Anderson Jr (1999))

Wind tunnel has been employed by engineers to improve the aerodynamic characteristics of vehicles since early 20’s. The first goal of wind tunnel testing is determining the shape of the vehicles to decrease the power that they need to move the vehicles at certain speed.

The importance of aerodynamic for the vehicles has been started from 1970. The goal is to reduce the fuel consumption by reducing the drag resistance. The most two factors to reduce the drag are:

- Reduce the frontal area of the vehicle to decrease the impact area of air.
- Aerodynamic improvement to reduce the drag coefficient (C_d).

1.2 General Information about the Study

Nowadays, the effort of automotive industries for vehicles production is to reduce the fuel consumption. Continues increase of fuel price, noise, and pollution are some problems that affect to improve vehicle aerodynamics for Manufacturers. Each vehicle to move constantly faced with the force of the fluid. In fluid dynamics drag is a force acting opposite to the relevant motion of any object moving respect to a surrounding fluid. Designers and manufacturers are trying to apply some modifications to reduce the drag force of the vehicle. There are some factors that can effect on vehicle drag such as, material of the body (some materials that can have less friction with the air), lines on the car body (Helping to cleave the air), and vehicle body design. For heavy commercial trucks, except aerodynamic design of the body manufacturers try to modify the body with different devices like, cab, air dam, aerodynamic mud flaps, side panels, hub caps, deflector, collar with roof fairing, tractor side panels and base bleed. To compare heavy commercial vehicles with other vehicles, they have been pointed low fuel efficiency, due to high aerodynamic drag. Researchers have been found that aerodynamic improvement for trucks is one of the most important technology when it comes to fuel saving. To overcome the aerodynamic force such as drag for typical large commercial vehicle at speed of 100 km/h, 50% of the total fuel to is allocated to provide the required power (Pevitt, Chowdury et al. 2012). Optimizations of fuel consumption for vehicles have been started between 1960-1970 decades. According to the previous researches on average, annual mileage has been changed between 130000 km and 160000 km for a typical heavy commercial vehicle's (Cooper 2004). Related to such a high mileage, slight changes in aerodynamic forces will effect on reduction of greenhouses gas emission and fuel savings. Odhams, Roebuck et al. (2010) have been found that 1.3 trillion litters of diesel and petrol consumed by heavy vehicles. From the burning of fossil fuels, it has been counted as a high level of pollution (CO_2).

1.3 Aim of the Study

The aim of this project is to reduces the drag force of the simple heavy commercial truck with trailer by different devices and see how much it is possible to decrease the drag. Although, in this project the importance of fillet edges instead of sharp edges for truck will discuss. At the end, air flow in the rear of the tractor and trailer and around them will discuss with vector lines and stream lines to show the vortex in the rear.

1.4 Solving the Problem Using Computational Fluid Dynamics

There are three distinguished main steps for every CFD problems such as: Pre-Processor, Solver and Post-Processor. At the beginning the problem has to import to the CFD software or code which is suitable for solving. Then, the problem will solve by numerical techniques and at the end in Post-Processor step it will enable to analyze the Problem and generate the data.

This project begins with research about modifying the shape of tractor and existing devices for tractor and trailer to reduce the drag. In this project the following steps have been employed to find the output for CFD Post-Processor:

- Generating the model by using SOLIDWORKS software
- Generating the unstructured mesh by CFX mesh software

- Boundary conditions have been employed For geometry
- Define the flow properties
- Transient flow were selected to be a natural condition
- Suitable time step has was selected
- Initial conditions were employed
- Drag Coefficient expressions were defined
- Solver was initiated

1.5 Organization of the Thesis

Unsteady models for various types of vehicles in different situation with experimental and numerical methods for previous works are presented in Chapter 2. The methodology used to solve the problem and the reason that why these conditions have been employed is discussed according to the solver output in post processing step of ANSYS-CFX® in Chapter 3. Chapter 4 contains the results of the output solver and the results have been presented inform of graphs and contours to show the results clearly for better illustration. Finally, conclusion of the thesis is written and the possibility for future work is explained in Chapter 5.

Chapter 2

LITERATURE REVIEW

Significance of Drag forces and Flow structure were explained in previous chapter. The purpose of following chapter is to give a well-illustrated literature review of the previous studies that have been done regarding to the problem at hand.

2.1 Introduction

Fluid flow analysis can be done by two ways such as experimental and empirical studies. There are many different methods to investigate the fluid flows characteristics. These purposes in new field of since that is called 'Fluid Mechanics'. Widely used of mathematics and differential equations and different results for different tests caused to capture theoretical and new equations. The two main method to present the fluid manner are including 1) Theoretical or numerical method and 2) experimental method.

Analyzing the result that has been obtained from experimental method was difficult to compute therefore, experimental method was just suitable to find the property of the flow previously. After passing some years, the equipment's have been improved for experimental test to be more accurate. Despite of all efforts some of the experiences remained the same problems. Because of some problem such as high cost, scale and environmental effects engineers have been start to find a solution to be cheaper and could solve the scale problems. In recent century improvement in computer science and programming brought the Computational Fluid Dynamics (CFD) to solve the numerical equations. This development helped the engineers to use simulations to find the changes in properties and analyze the fluid flow in different conditions for internal and external flow. Moreover, this method is cheaper than experimental methods for analytical problems.

2.2 Vehicle Aerodynamics

Nowadays, mechanisms of drag reduction for trucks have been equipped. There are lots of studies about experimental and numerical analysis about heavy vehicles aerodynamic. Most of the research in these days has been focused on drag reduction for trucks with new design (Schoon 2007, Corin, He et al. 2008, Mohamed-Kassim and Filippone 2010). Mohamed-Kassim and Filippone (2010)has been employed several aerodynamic retrofitting techniques for fuel consumption and to reduce the drag for heavy vehicles.Lenngren and Håkansson (2010) explored the influence of aerodynamically shaped and trailer devices by computational fluid dynamics (CFD). Miralbes and Castejon (2012) have been made a comparison between aerodynamics of boat tails on heavy trucks to reduce the drag coefficient with a vehicle without the given device.McCallen, Flowers et al. (2000)developed a method for an integrated tractor-trailer model to reduce the aerodynamic drag of heavy vehicles by experiment on aerodynamic flow analysis and numerical simulation. They have been developed an advanced computational models that utilize an LES approach, in addition to the use of state-of-the-art RANS modeling. Englar (2001)used a wind tunnel test on a generic truck model to present the effect of the gap between the tractor and trailer.Roy and Srinivasan (2000) present the aerodynamic of vehicles with high-sided and normal trucks to decrease accidents on the road due to wind loading and decrease the fuel consumption. They have been focused on exterior rear-view mirrors drag. Although, they have been stated that modification of the truck geometry can effect on the fuel consumption and drag reduction. Experimental analysis for a

Ground Research Vehicle (GRV) to present the base drag on the large scale vehicles at subsonic speed has been employed(Diebler and Smith 2004). They focused on base drag of trucks, motor homes, buses, reentry vehicles, and other large-scale vehicles. They presented preliminary results of both the effort to define a new base drag model and the investigation concerning a method to reduce the total drag by manipulating for body drag.Kamal and Yamin (2006)have been made simulation on external flow analysis of a coach with computational fluid dynamics (CFD) techniques. Their given results have been presented that steady state CFD simulation can be used to enhance the aerodynamic development of a coach.Aziz and Gawad (2006) have been founded the effect of front shape on the characteristics of the flow field and the heat transfer numerically and experimentally that is produce in the rear of the buses in driving tunnels. They made three different models with flat-, inclined-, and curved-front shapes. Although, they explored that Stability of the buses in driving tunnel condition can be the effect of front shape of them. Also, they found that curved-, and inclined-front shape for vehicles can be better than flat-front about 20% in cooling system. François, Delnero et al. (2009) studied about a specific type of bus in the Argentinean routes with cross-wind condition. They had experimental analysis about aerodynamics characteristics and response of a double deck bus. In addition, pressure distribution of the frontal and lateral section of the vehicle together with Aerodynamic forces corresponding to the CG of the bus have been measured. Mohamed-Kassim and Filippone (2010)analyzed potential of fuel-saving with the effect of devices on heavy vehicles for drag reduction. Their results have been presented that the performances of these devices are better in long-distance routes than urban areas. Also, they mentioned that on the long distance routes generally save twice as much fuel as short distance routes.Katz and Dykstra (1992)have been presented that there are three general parameters that effect on the aerodynamics of the vehicles that is shown in Figure 2.1. These three parameters are including side-slip angle, ride height, and body's incidence. Also, these parameters depend on the vehicle's geometry. They also present that changing the attitude of the most vehicles are similar to wings, Figure 2.2 has been shown that with increasing of the angle of attack, down force will increase.

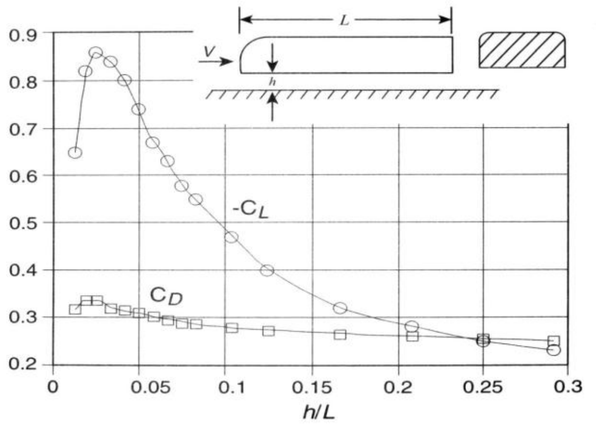


Figure 2.1. Lift and Drag coefficient versus ground clearance (taken from Katz and Dykstra (1992))

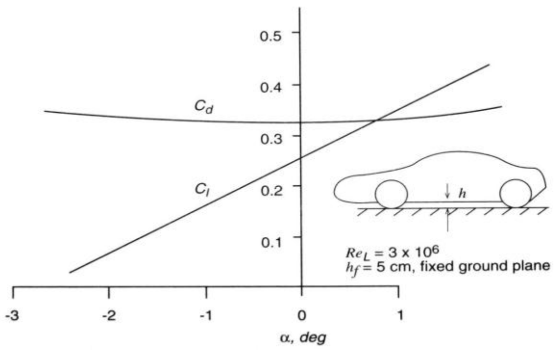


Figure 2.2. Lift and Drag coefficients versus Angle of Attack (taken from (Katz and Dykstra 1992))

Effect of aerodynamic drag on the typical passenger vehicles can be as high as 65% of the total drag at a normal speed of 100 km/h (Yang, Schenkel et al. 2003). The accuracy of the well simulation computational fluid dynamics (CFD) analysis is within 6-8% on the drag coefficient of the vehicle with predicted aerodynamic drag (Yang and Schenkel 2004).Hucho (1998) presented that in the different wind tunnel tests the difference in drag coefficient values can be up to 5%. The atmospheric turbulence intensity commonly is on the range of 2% and 10% according to the measurements that have been presented by(Watkins, Saunders et al. 1995).

The turbulence intensities can be as high as 20% when there are two vehicles in tandem with distance of three typical car length to each other (Saunders and Mansour 2000).Haruna, Nouzawa et al. (1990) have been found that the stream wise vortices are a source of aerodynamic noise for a flow behind a three dimensional bluff body.Duell and George (1999) have been worked on the three dimensional bluff body with a blunt base to present the flow result near wake regions. Average drag coefficient (Cd) values have been decreased by the time from 0.7 for the old vehicles with boxy design shapes to 0.3 for new design of the vehicles to be more streamlined (Desai, Channiwala et al. 2008).Singh, Rai et al. (2005)have been illustrated that for the vehicle in moving situations the effect of drag is proportional to the square of velocity, so with increase in velocity (at approximately 50 km/h), aerodynamic drag becomes one of the most eminent factors contributing to the total drag experienced by the vehicle. Computational analysis for drag reduction has been employed by (Rouméas, Gilliéron et al. 2009, Barbut and Negrus 2011) on road vehicles and for simplified car body (Ahmed body) Guilmineau (2008) has been performed. Because of complex geometry for vehicles Ahmed, Ramm et al. (1984) have been presented a simplified vehicle geometry with fully three dimensional regions of separated flow for better illustration of such flows. Ahmed's body's dimensions have been measured 1044 mm in length, 288 mm in height and 389 mm in width. The distance between ground and bottom surface of Ahmad body is 50mm the geometry has been shown in Figure 2.3.

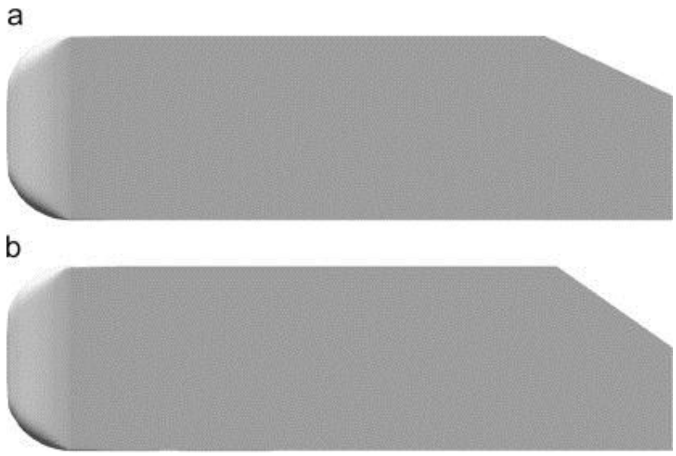


Figure 2.3. Ahmed body view: (a) 25° rear slant; (b) 35° rear slant (Taken from Guilmineau (2008)).

According to the introduction of this chapter, comparison between CFD analysis and experiments are the strong reason to use the new technology as a requirement of each research to minimize the restriction of CFD. Therefore, in this study numerical simulation with ANSYS-CFX to reduce the aerodynamic drag for truck-trailer is performed.

Chapter 3

METHODOLOGY

3.1 Introduction

In this chapter, the procedure of this study is explained. As it is presented in the first Chapter, this project has been operated by evaluating different drag reduction devices for truck-trailer by means of Computational Fluid Dynamics (CFD).The numerical study was carried out to find the effect of aerodynamic devices on heavy commercial vehicles by applying Computational fluid dynamics. The quantitative investigation has been performed on the Truck and its Trailer in three dimensional flows by means of finite volume method. Control volume technique has been applied to change the partial differential equation format to the algebraic equations and fined a capable solution to satisfy governing equations in each single element of the grid. In this study visualization of CFD have been used to show different graphs and charts to analyze and compare the fluid flow structure and the vortex in the rear of trailer and the empty space between truck and trailer. Although, the difference between each device and the effect of them on the Drag will be discuss. This project has been performed in three steps: pre-processing by SOLIDWORKS and ANSYS/CFX® for generation of geometry and code generation respectively, processing with ANSYS/CFX® and at the end post processing by ANSYS/CFX®. All of these processes will explain in this chapter step by step in detail.

3.2 Pre-Processing

The simulation has been performed by ANSYS/CFX® and each simulation runs for 4000 iterations because, as we defined a transient simulation the total time and time steps has been defined as 0.4 s and 0.0001s respectively. To make sure that the solutions are acceptable, the convergence has been checked from drag-plot and the residuals. The Convergence of the residuals has been presented in Figure 8.

3.2.1 Geometry Modeling

In this project the geometry has been generated by SOLIDWORKS commercial program to generate a three dimensional model and imported to the ANSYS/CFX® to analyze it. The dimensions of the simplify truck has been taken from the Mercedes Benz official website and for the trailer the dimensions have been taken from VEHICLELOAD ANDSIZE LIMITSGUIDE 2005, To be a real dimension to present the most accurate solutions and analysis. After importing of the geometry to ANSYS/CFX® Enclosure and Boolean has been created to visualize the real environment for the truck and trailer in moving condition with speed of 25 m/s respectively. The total length of the tunnel is seven times of the length of the truck (7*L) and the height and the width are twice of the truck and the trailer length (2*L).Although, in this study to reduce the time consumption of the solution the geometry has been divided symmetry to reduce the width of the truck and trailer. The model has been shown in Figure 3.1. The dimensions of the truck-trailer have been presented in the Appendix A.

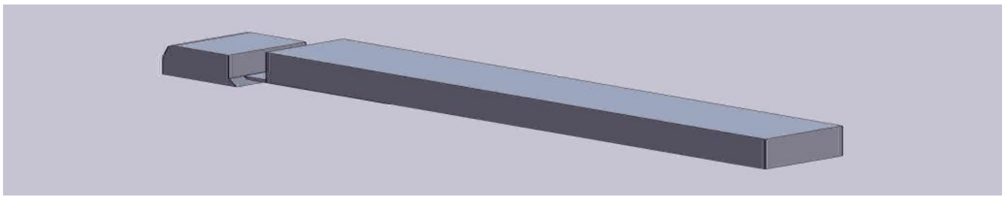


Figure 3.1. Isometric view of the simple model

3.2.2 Meshing

After finishing the geometry procedure and defined the name selections, meshing with CFX software has been started. The x-axis has been considered to be the direction of airflow. At the beginning whole geometry should have the combination of Curvature and Proximity meshes for unstructured body with fine relevance of center. These options have been help to have a fine mesh all around the geometry. After that, to have a better mesh around the tractor and trailer the Inflation has been used to make 10 layers with growing rate of 1.2 for first layer thickness option. In the regions with unsteady wakes, like the base of trailer, a large number of cells and nodes have been employed to show the vortices and improve the resolution and robustness of the total solution. Because of limitation in the time, volume meshes around the truck and its trailer have been considered between 60-61 thousand cells. Picture of the meshing with inflation around the tractor and trailer is shown in Figure 3.2.

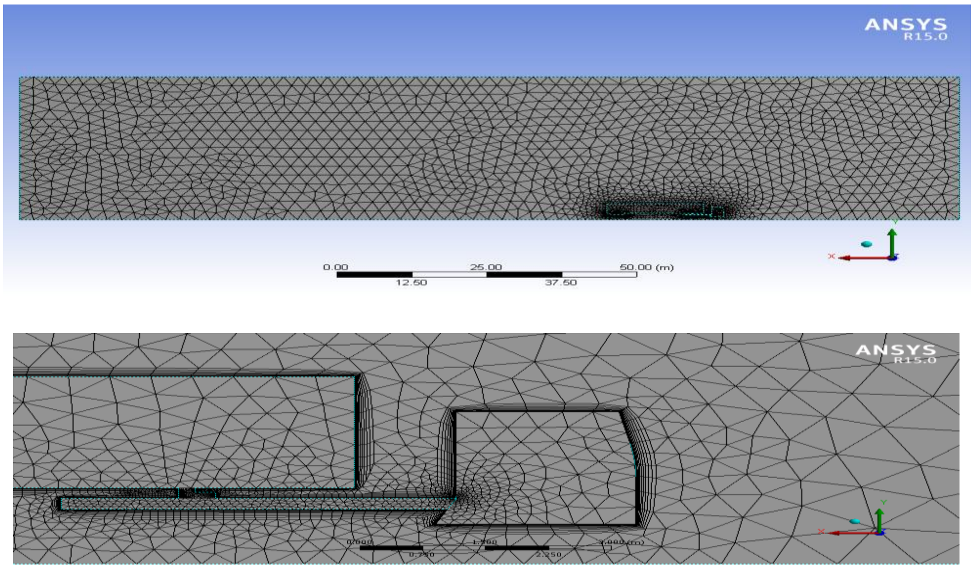


Figure 3.2. Volume mesh of the computational domain and the inflation layers

3.2.3 Physical models

In below a brief explanation of the most required models that has been used for simulation for ANSYS-CFX® presented.

- **Space** - Three dimensional
- **Motion** - Stationary
- **Time** - Unsteady
- **Material** - Air at 25^{o}C
- **Viscous Regime** - Turbulent

  - **Model**–Reynolds Averaged Navier Stokes - k-ε turbulence model

3.2.4 Case Study

In this project, there have been five selected devices such as Deflector, Gap Fairing, Vortex Stabilizer, Frame Extension, and Teardrop have been employed for trailer and replacing smooth edges instead of sharp edges has been used for truck that they are presented in Table 3.1. In this section each case description has been explained separately. The devices information received through a research of today's market, however without drawing. The design of these devices and some own design are performed by SOLIDWORKS software by the author. Although, the dimensions of the truck and trailer and the devices are shown in Appendix B.

Table 3.1.Selected devices for truck-trailer modifications

Gap treatment	Deflector, Vortex stabilizer, gap fairing
Base treatment	Frame extension
Aerodynamic trailer designs	Teardrop
Aerodynamic truck designs	Smooth edges

Gap Treatment The gap treatment devices are presented in Figure 3.3 to Figure 3.5. Deflector is one of the most important devices to avoid flow distribution to enter between the gaps. This device can create an upstream of air to sweep up and over the heavy commercial vehicle.

Vortex stabilizer includes two flat plates next to each other with a specific distance in front of the trailer. The objective of this device is to decrease disturbance from the cross winds and create stable vortices between tractor and trailer.

Gap fairing is the device to reduce the pressure peak at the edges and upper corners by re-attachment of the flow on the trailer, however it has been set up in front of the trailer.

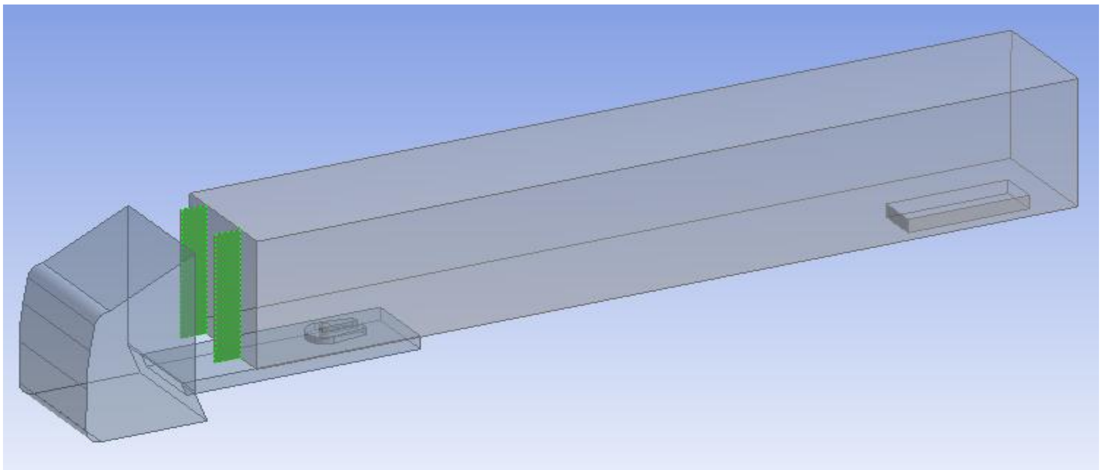


Figure 3.3. Vortex Stabilizer

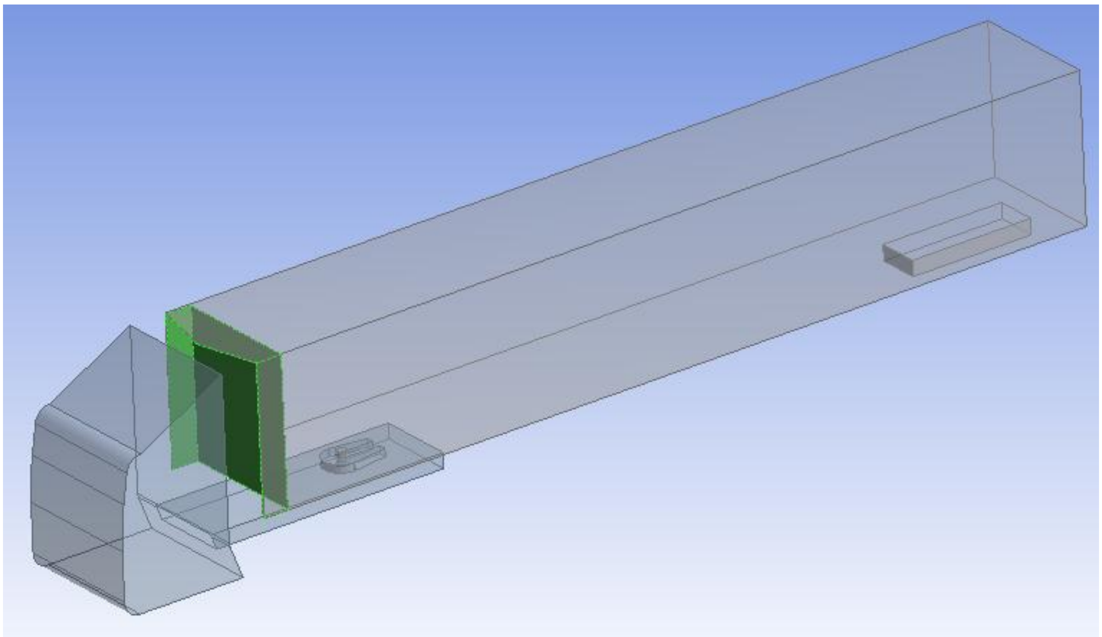


Figure 3.4. Gap Fairing

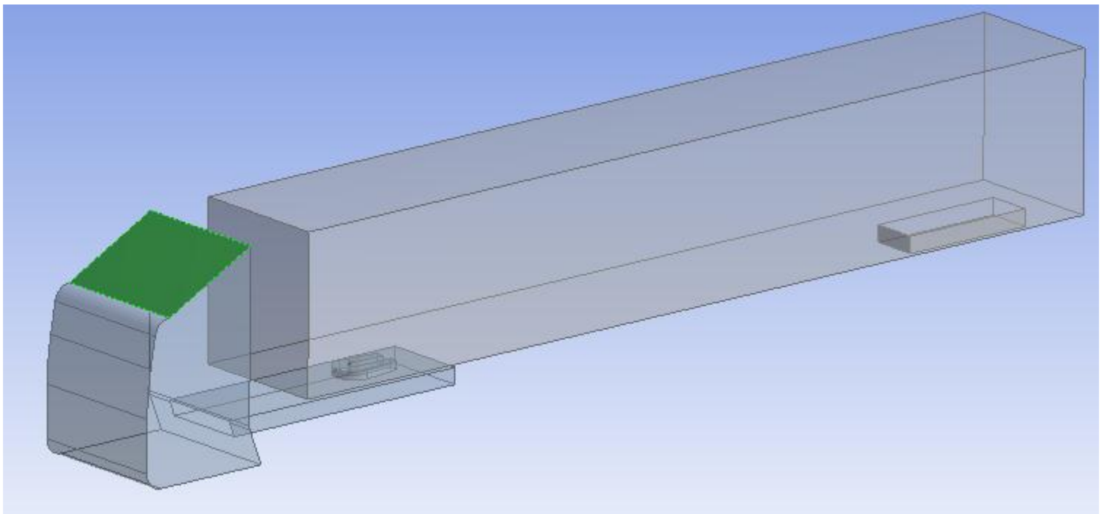


Figure 3.5. Deflector

Base Treatment The base treatment devises are shown in Figure 3.6.Frame extension is the combination of base plates and side plates. These plates could decrease the drag by decrease the base wake. McCallen, Salari et al. (2005) have been performed that with inward angle of 13^{o} and the length of 1/4 of the trailer width it can achieve to the optimal point for drag reduction.

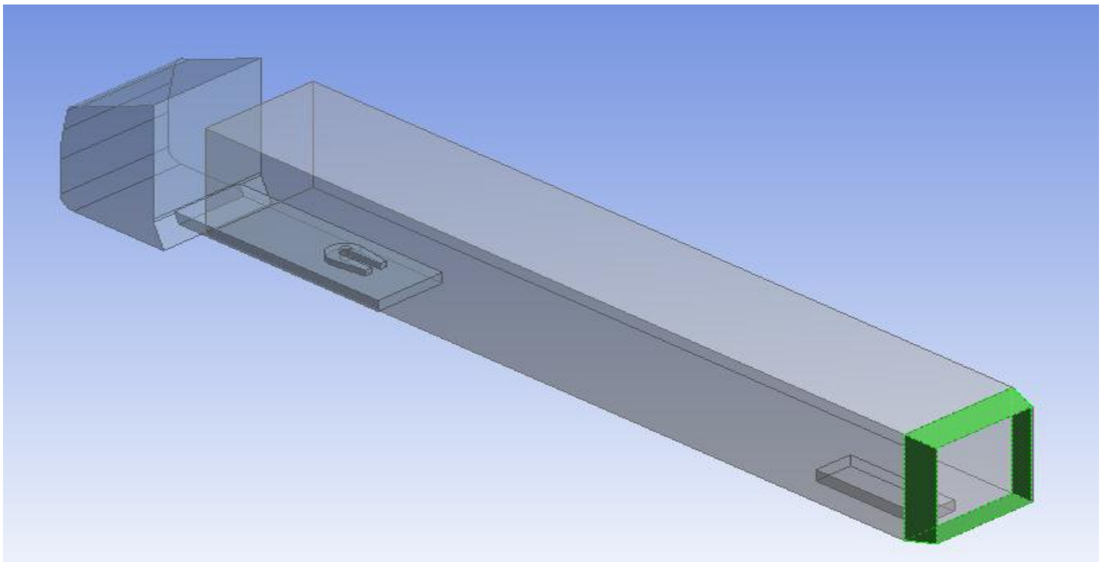


Figure 3.6. Frame Extension

Aerodynamic trailer designs Teardrop is one of the modifications for trailer as it is shown in Figure 3.7. This device has been used to improve the flow over the truck and decrease the rear wake. Due to the limitation of the height in USA the height of the Heavy commercial vehicles should not be more than 4.27 meter therefore according to the trailer height the height of the teardrop has been assumed as 0.5 meter.

Aerodynamic truck designs. In this project instead of using sharp edges in front of tractor the smooth edges have been shown to analyze the amount of drag reduction and its effects on the flow.

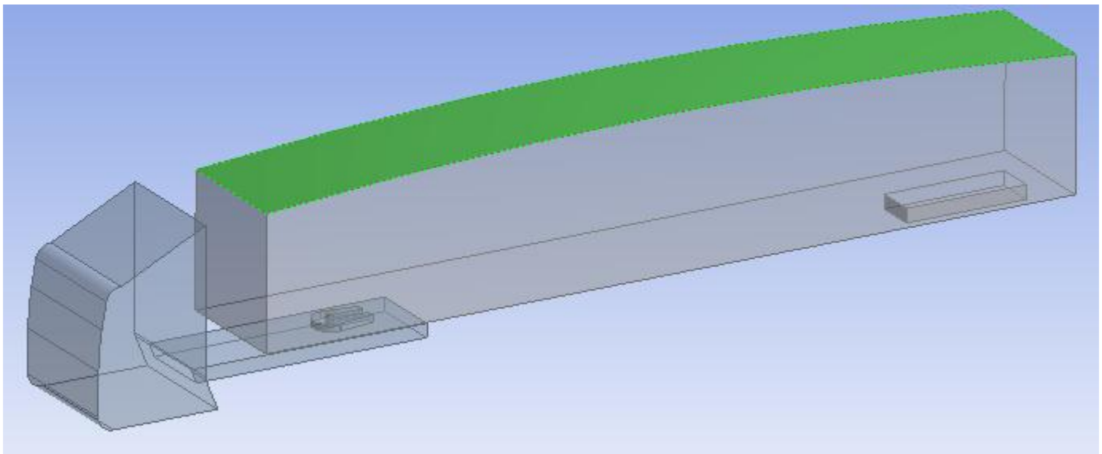


Figure 3.7. Teardrop

3.2.5 Boundary Conditions

According to the circumstance and the driving conditions, the trucks with its trailer have been subjected to wind with axial direction (x-direction). In this project wind tunnel have been divided into several parts such as, Inlet, Outlet, far field, Sky, Outlet, Ground, and Symmetry as it has been shown in Figure 3.3.

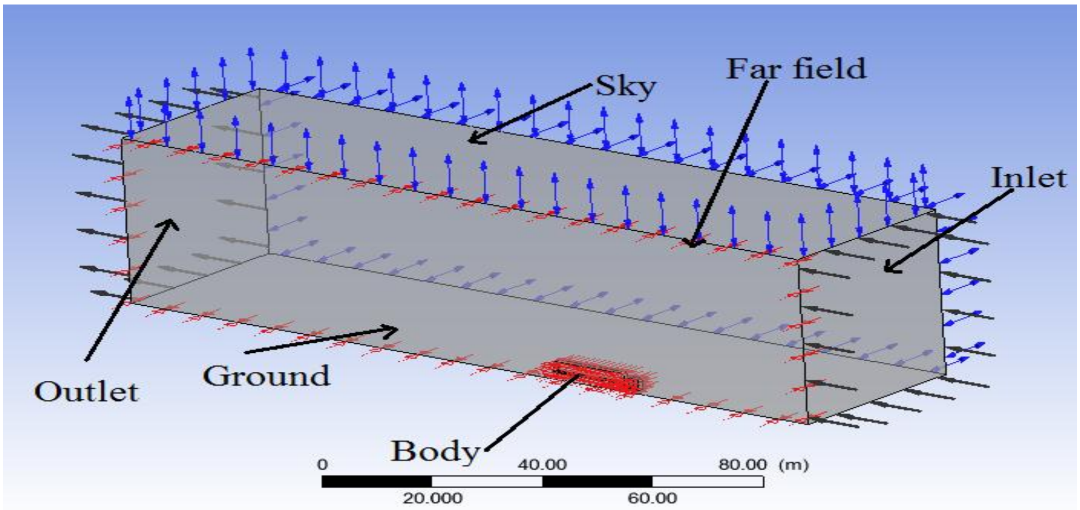


Figure 3.8. Wind Tunnel Boundary Condition ANSYS/CFX®

The yaw angle that has been used in the simulations is 0^o and for the 0^o yaw angle, inlet is set to velocity inlet. However, Sky and Far field has been defined as an open environment to make the simulation in real conditions. To simulate the rigid parts the tractor and trailer parts has been set as no slip condition. To show that the body is in moving condition, the ground has been defined as no slip wall with velocity of 25 m/s in U-direction and zero for the V and W directions. Finally, the rest of the boundary conditions are shown in Table 3.2.

Table 3.2. Boundary Conditions of the Simulation

Tractor	Stationary wall- no slip	-
Trailer	Stationary wall- no slip	-
Tunnel Inlet	Velocity Inlet	25 m/s
	Turbulent Intensity	5%
Tunnel Outlet	Relative Pressure	0 Pa
	pres. Profile Blend	0.05
Tunnel Ground	Moving wall in x-direction	25 m/s
	Wall roughness - Smooth wall	-
Tunnel Roof	Relative Pressure	0 Pa
	Opening - Entertainment	-
	Turbulent - Zero Gradient	-
Tunnel side wall	Relative Pressure	0 Pa
	Opening - Entertainment	-
	Turbulent - Zero Gradient	-
Symmetry	Symmetry	-

3.3 Aerodynamic Drag Calculation

The drag coefficient (Cd) is useful to compare different vehicles aerodynamic efficiency. However, drag coefficient is related to vehicle speed, frontal area, drag force, and the density as it shown in equation 1.

$$C_d = \frac{F_d}{1/_2 * \rho * A * V^2} \quad (1)$$

Where, F_d, ρ, A, and V are representing Drag force, Density of air, frontal area and velocity of the truck respectively.

Drag Force has been founded by processor of the ANSYS-CFX® as a force in x direction on the truck body. After that, by writing the expressions in post processing to find the drag coefficient.

The term in below is called as Dynamic Pressure as a first expression:

$$DP = \frac{1}{2} * \rho * V^2 \quad (2)$$

Where, ρ and V represent the density of the air and velocity of the truck respectively.

Then Frontal Area has been calculated as:

$$A = h * w \quad (3)$$

Where, h and w represent the height and width of the truck-trailer.

At the end after bringing all of these terms to each other the drag coefficient has been calculated for the current geometry.

Chapter 4

RESULTS

4.1 Introduction

In the following chapter use of CFD analysis to show the effect of tractor and trailer devices and modifications on the truck to reduce the drag of the heavy commercial vehicles and presenting the flow structure are completed in the following way:

- Analysis Results
- Truck Aerodynamic Modification
- Results of individual devices
- Final Remarks

4.2 Results

In this project ANSYS-CFX 15.0® has been used to generate the results in different contour forms to provide better comprehension of the flow structure. In this research 6 different analysis and devices are employed to the Truck and the Trailer for drag reduction. As an ultimate aim of this study 41% reduction in drag has been achieved. All of the simulations have been examined by k –ε method and as a most outcomes, results from graphs are represented below.

4.3 Truck Aerodynamic Modification

In this section Truck has been modified by changing the sharp edges in front to smooth edges by using filet and show the effect on the pressure, velocity, and the drag.

4.3.1 Drag Analysis

At the beginning Truck has been simulated without any smooth edges to measure the drag of it and compare it with different smooth edges. After that three different simulations has been employed to see the effect of each smooth edges on the drag as they are shown in Figures 4.1 to 4.4.

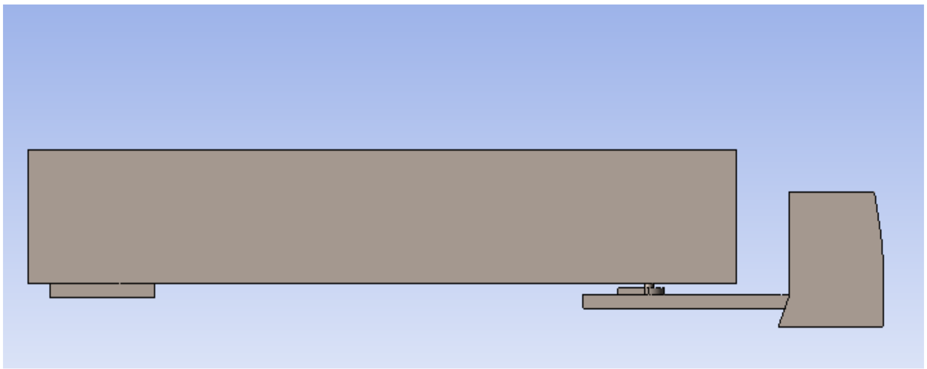


Figure 4.1. Truck model with no aerodynamic modification

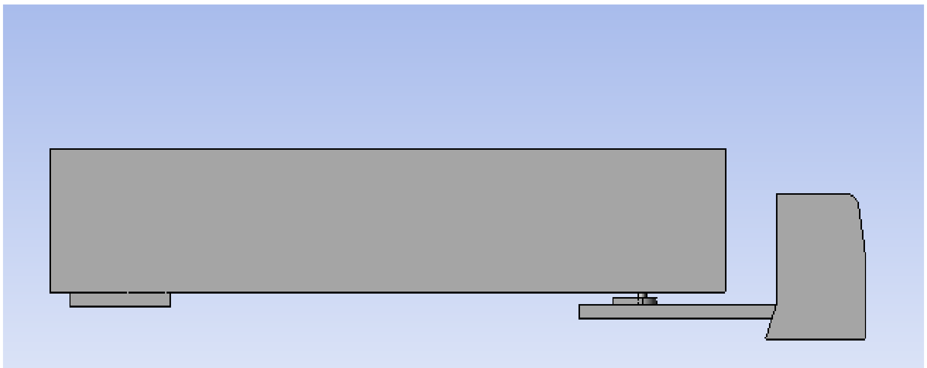


Figure 4.2. Truck model with top filet edge

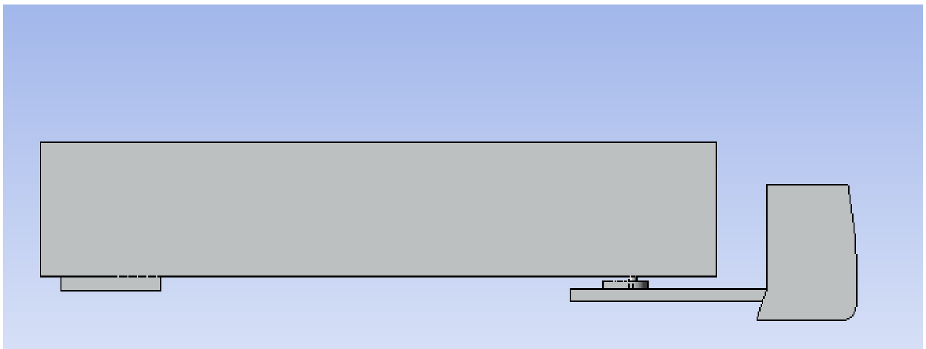


Figure 4.3. Truck model with bottom filet edge

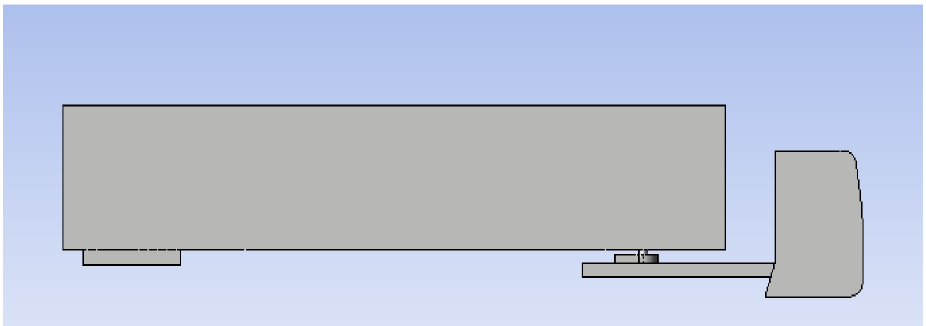


Figure 4.4. Truck model with top and bottom filet edges

According to the modifications on the truck body, it has been found that the drag coefficient has been reduced gradually. For the first case (Without any filet edges) the drag coefficient has been calculated as 1.17. For the other three modification results are given in Table 4.1 to compare the drag of all modifications to each other to find the best choice for truck.

Table 4.1. Drag reduction in presence of fillet edges

Model	Cd	Reduction
Without Any Filet Edges	1.17	-
Top Filet Edge	0.71	39%
Bottom Filet Edge	0.97	17%
Top and Bottom Filet Edges	0.95	18%

4.3.2 Aerodynamic Analysis

As it has been shown in Figures 4.5 to 4.8 pressure distribution in front of the truck hasn't been affected by making the edges smoother. On the other hand, they have been affected on the pressure between the cab and trailer

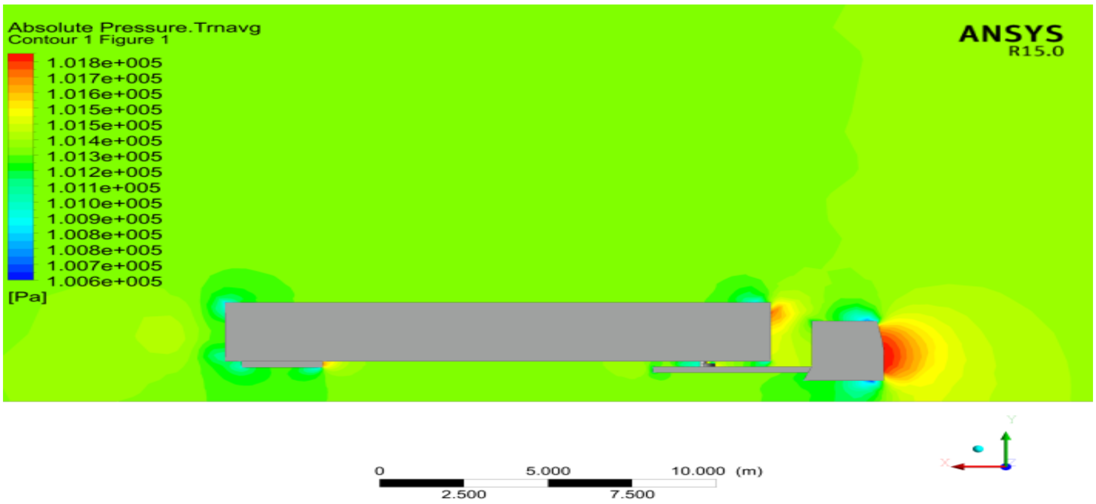


Figure 4.5. Pressure Contour for model with no aerodynamic modification

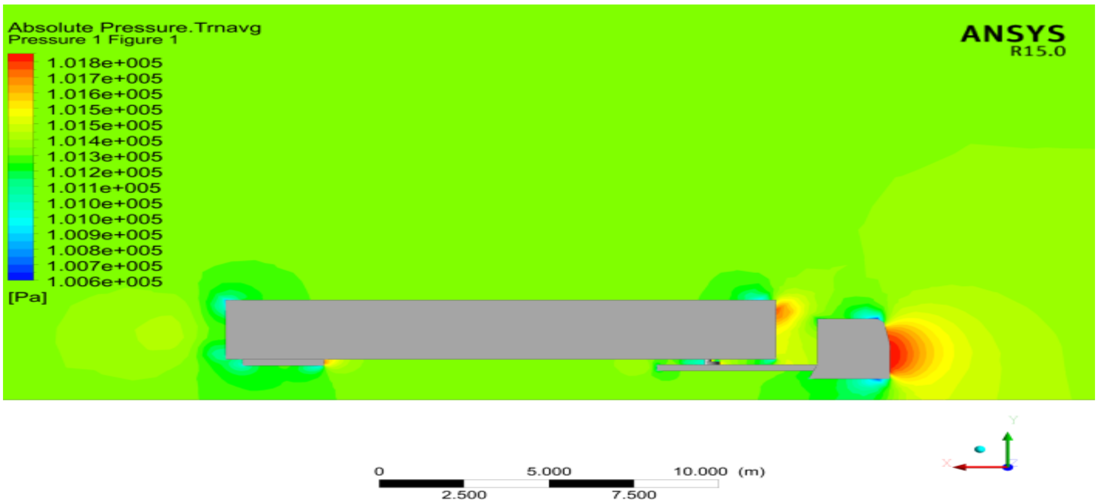


Figure 4.6.Pressure Contour for model with top filet edge

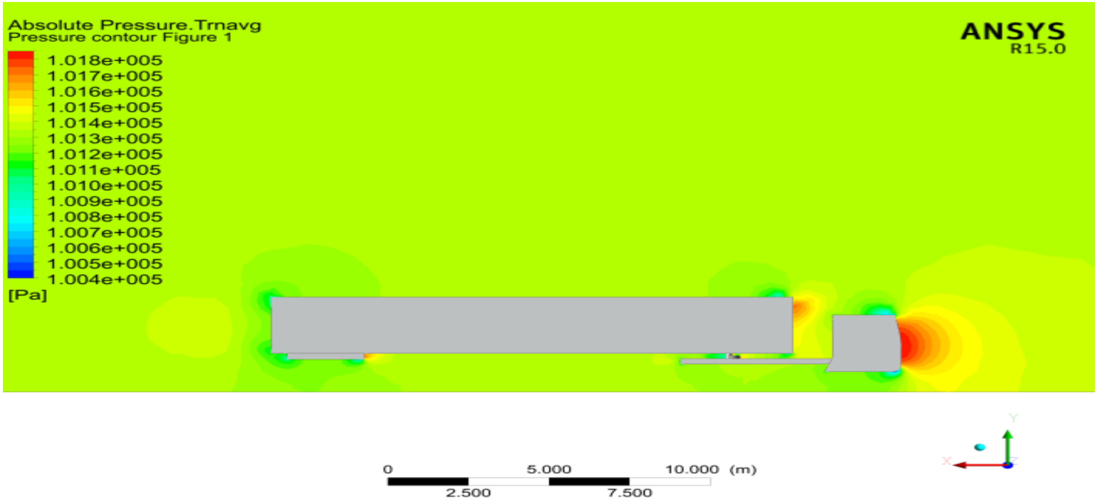


Figure 4.7. Pressure Contour for model with bottom filet edge

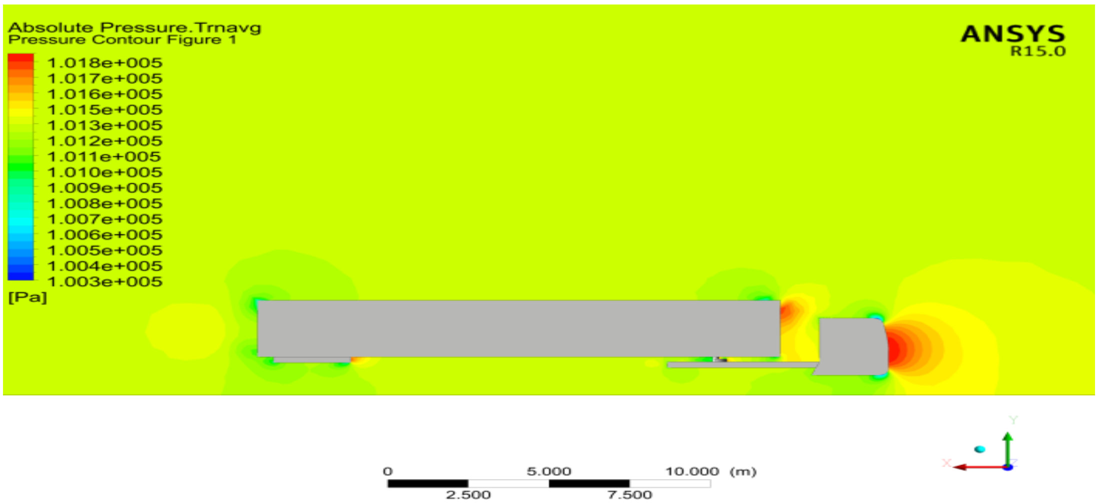


Figure 4.8. Pressure Contour for model with top and bottom filet edge

Two low-pressure bubbles are observed in Figure 4.8oneof them is on the roof of the truck and the other one has shown on the bottom of tractor fore. Because of the rounded edges the flow acceleration will increase and the low-Pressure bubble on the cab will attached to the surface.

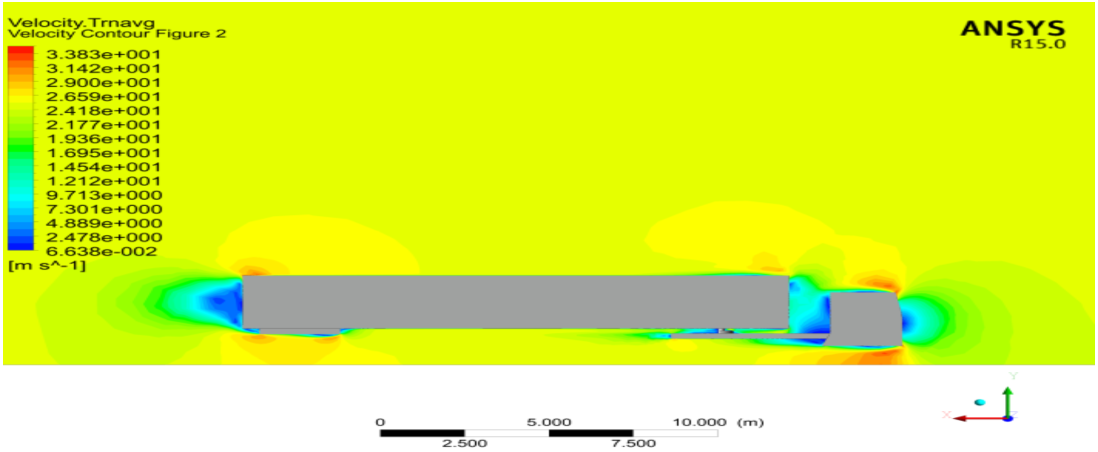


Figure 4.9. Velocity Contour for model with no aerodynamic modification

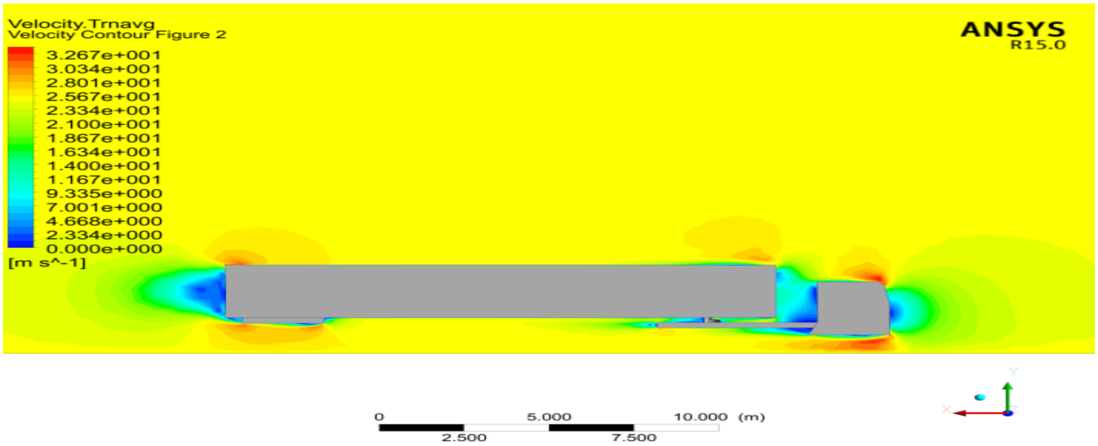


Figure 4.10. Velocity Contour for model with top filet edges

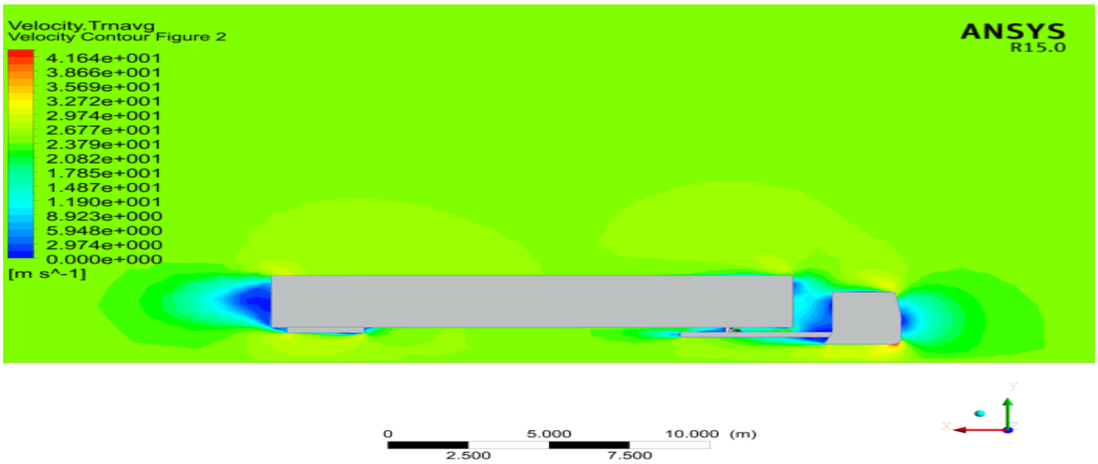


Figure 4.11. Velocity Contour for model with bottom filet edges

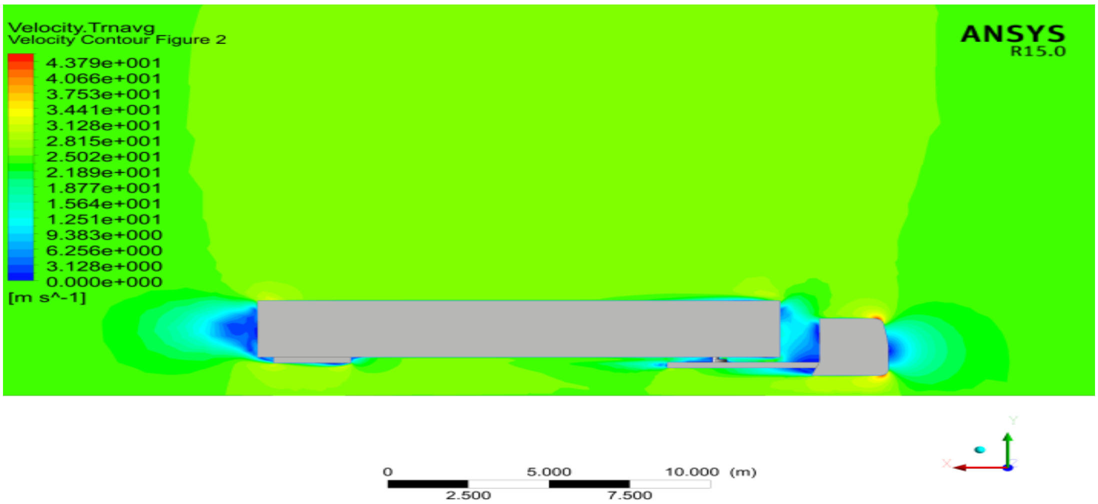


Figure 4.12. Velocity Contour for model with top and bottom filet edges

As it has been shown in Figures 4.9 to 4.12 filet edges in front of the truck have been effected on the velocity between the cab and trailer. From the top filet it can illustrate that the velocity on the top of the trailer in front and in the rear will increase. On the other hand, bottom filet and both filet edges will increase the area of the vortex to make a vacuumed between the cab and trailer. According to this reason the C_d of the second case (Top filet edge) is less than the rest of the cases.

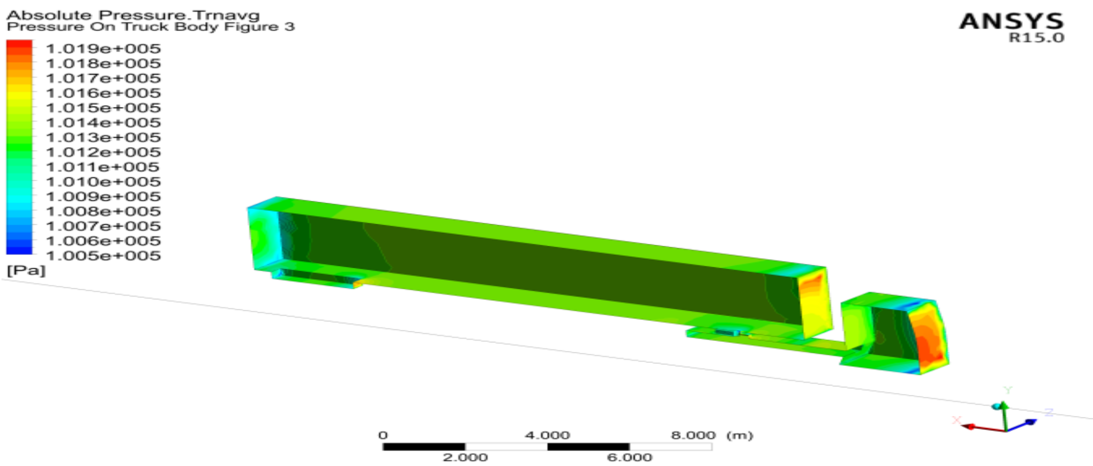


Figure 4.13. Pressure distribution contour on truck body for model with no filet edges

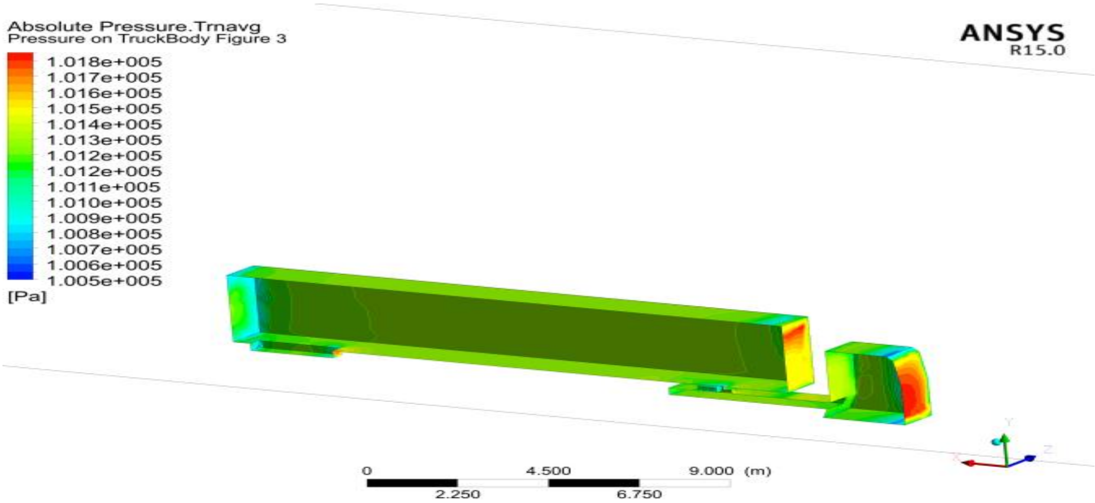


Figure 4.14. Pressure distribution contour on truck body for model with top filet edges

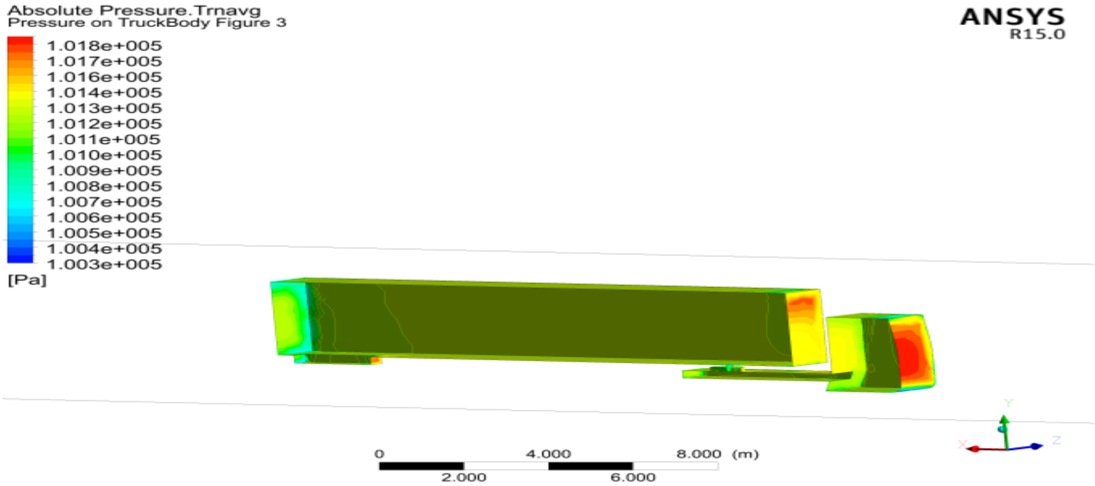


Figure 4.15. Pressure distribution contour on truck body for model with bottom filet edges

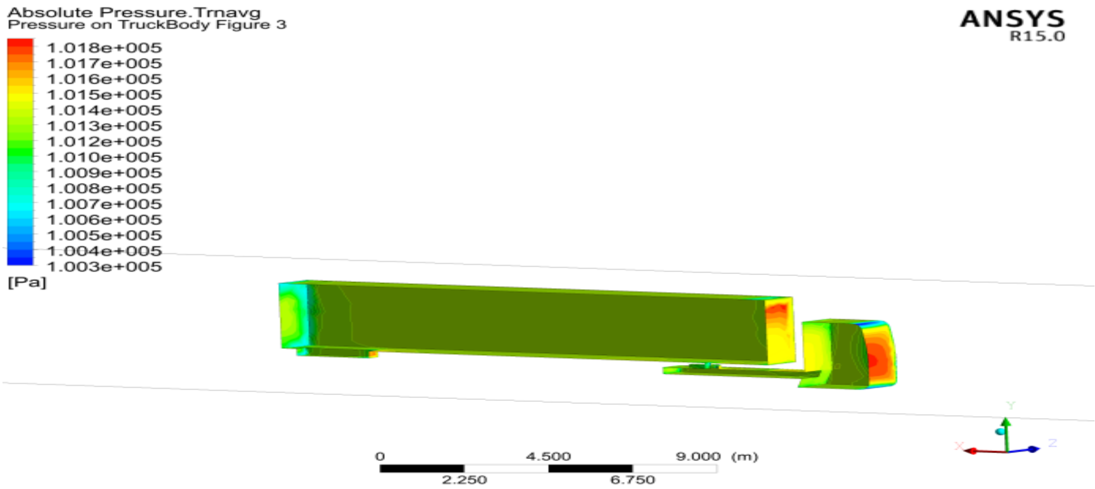


Figure 4.16. Pressure distribution contour on truck body for model with top and bottom filet edges

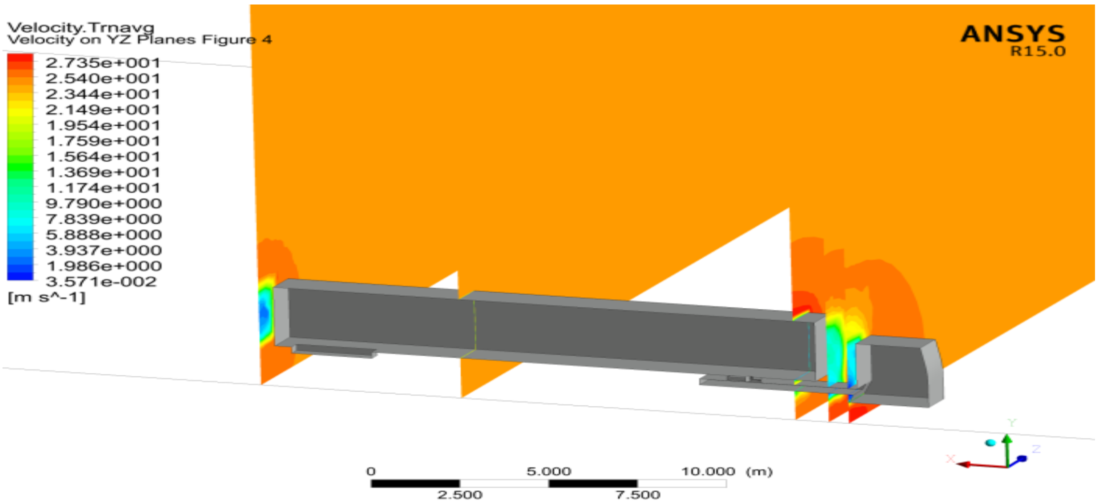


Figure 4.17. Velocity contour on planes for model without any filet edge

From Figures 4.13 to 4.16 it has been illustrated that filet edges will not effect on the pressure distribution in front of the truck. In fact, with top filet edge the pressure in front of the trailer on top, the area with pressure $1.018*10^5$ increased more than other cases.

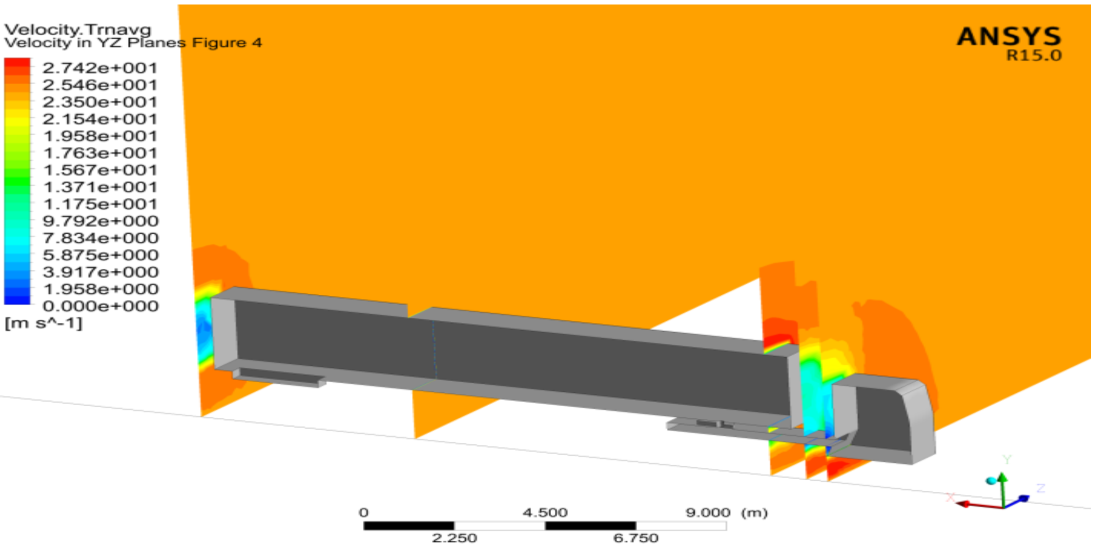


Figure 4.18. Velocity contour on planes for model with top filet edge

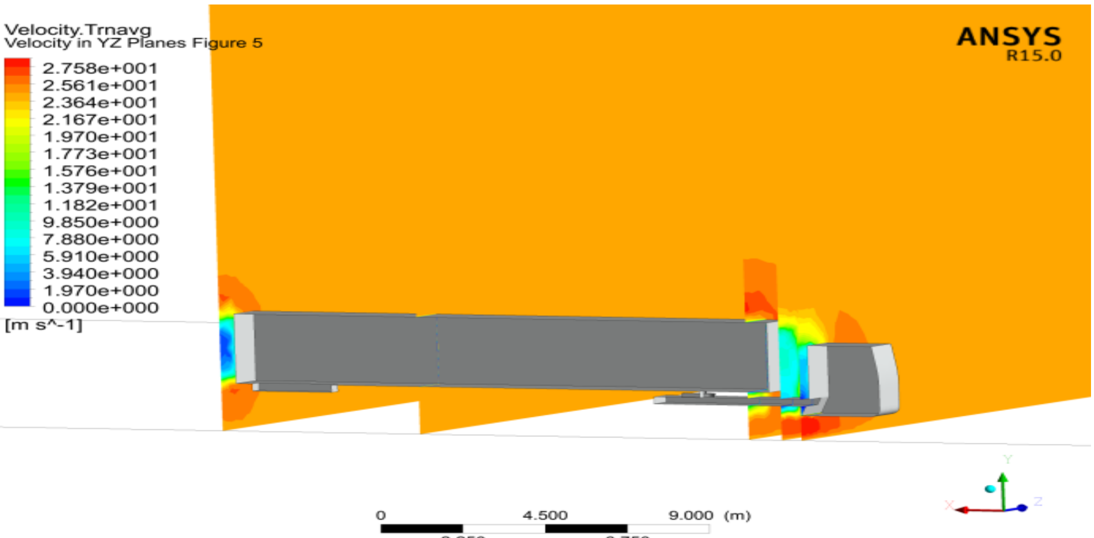


Figure 4.19. Velocity contour on planes for model with bottom filet edge

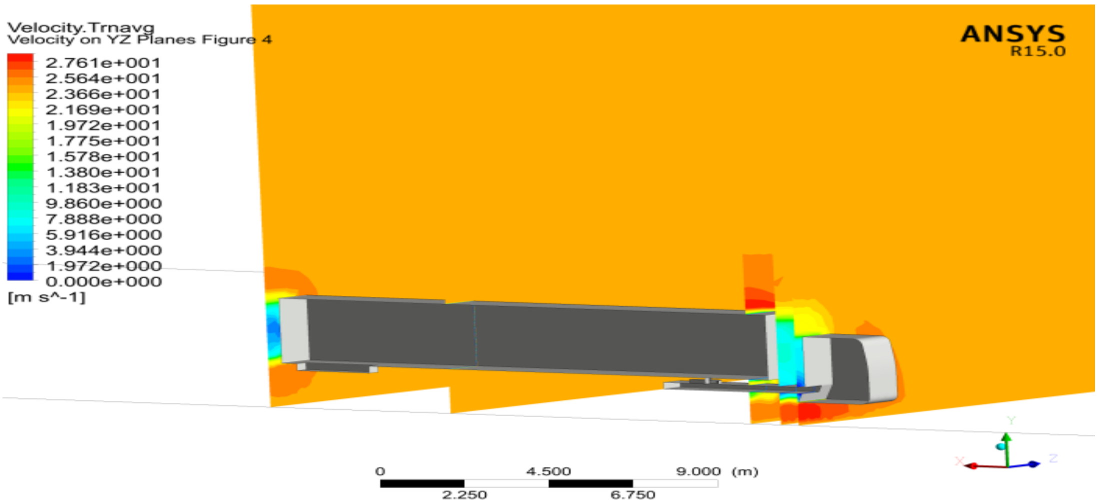


Figure 4.20. Velocity contour on planes for model with top and bottom filet edge

Filet edges can affect the velocity of the flow in the rare of the trailer and free space between the cab and trailer as they have been presented in the Figures 4.17 to 4.20. From Figures 4.19 and 4.20 to compare these cases together it has been illustrated that with bottom filet the area that velocity is almost 0 m/s in case with bottom filet edge is more than case with top and bottom filet edge although, it will make a vacuum in the rear of the trailer and then it will increase the aerodynamic drag.

4.4 Results of individual devices used for aerodynamic drag Analysis

In this section the effect of aerodynamic devices on the drag and the pressure and velocity analysis have been presented. There are five different devices installation on the truck and its trailer such as: Deflector, Teardrop, Base treatment, and Gap treatment the results have been shown in Figure 4.21.

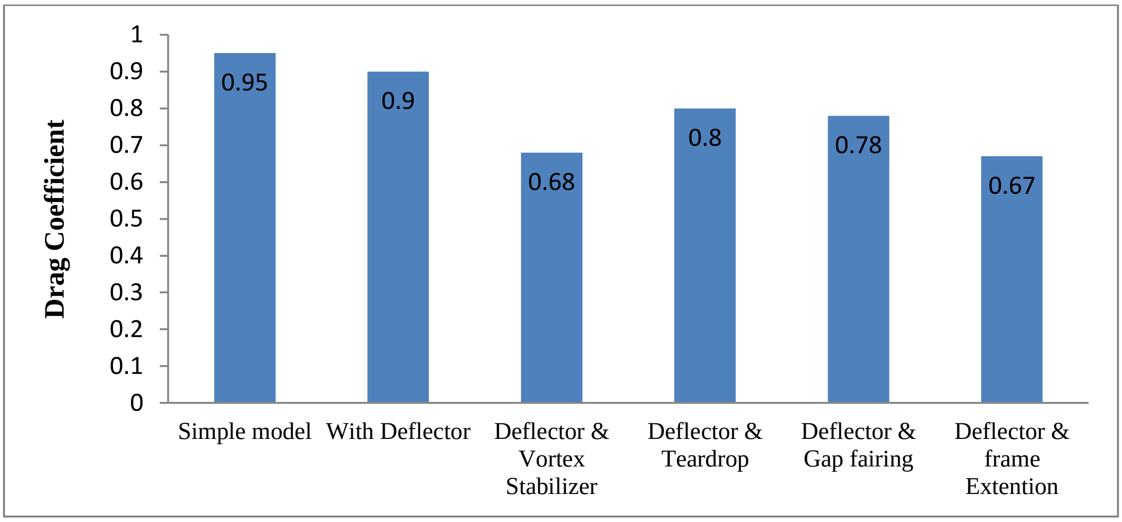


Figure 4.21. The results of the individual devices for truck and trailer

4.4.1 Deflector:

Flow streamline over the truck as demonstrated in Figure 4.24, reveals that flow can pass the gap to the top of the trailer and Table 4.2 has been shown that deflector can decrease the amount of drag to 0.9 that is 5% improvement on aerodynamic drag.

Table 4.2. Change in drag with deflector

Model	Cd	Reduction
With Deflector	0.9	5%

By presenting the velocity vector in XZ plane for effecting of the deflector on the gap, Figures 4.22 and 4.23, it is obvious that without using deflector the flow is very irregular between cab and trailer.

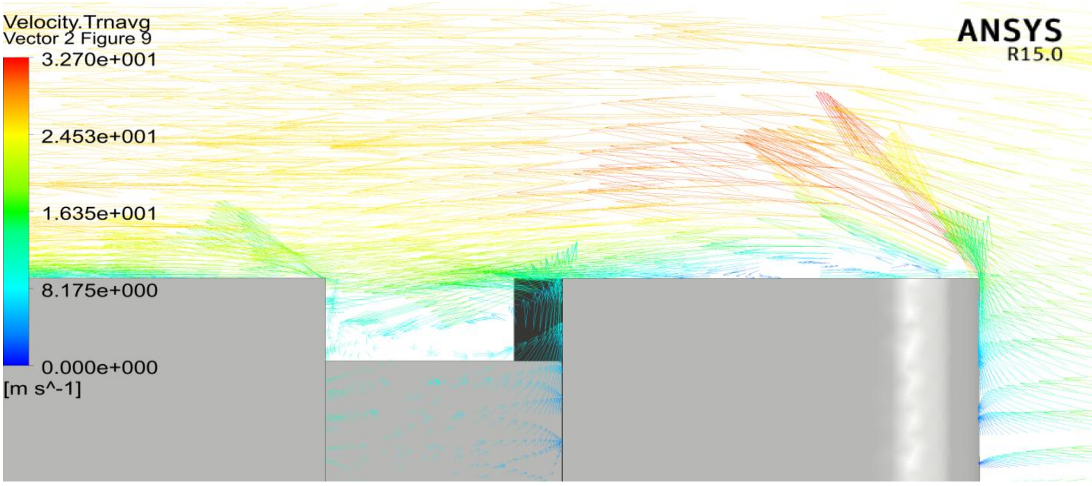


Figure 4.22. Velocity vector on XZ plane between cab and trailer for model with top and bottom filet edges

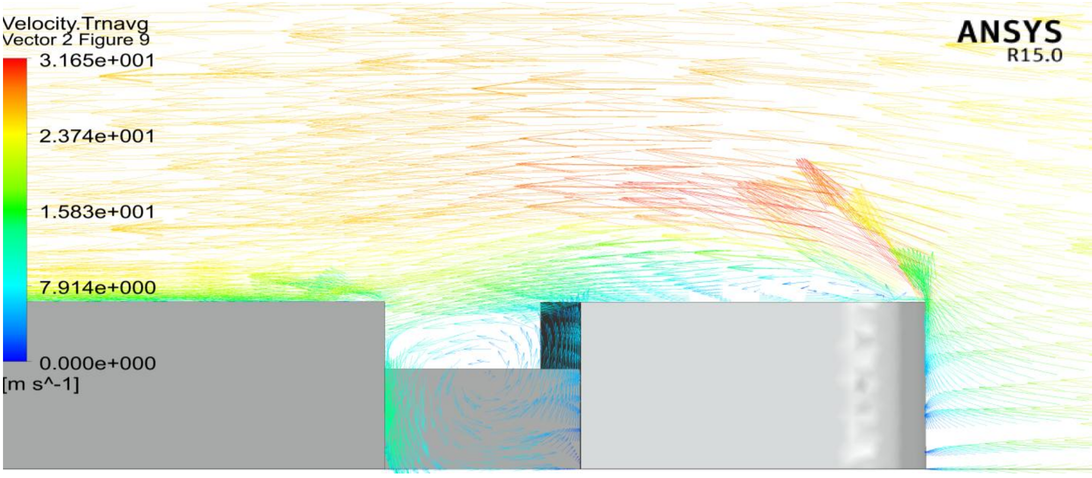
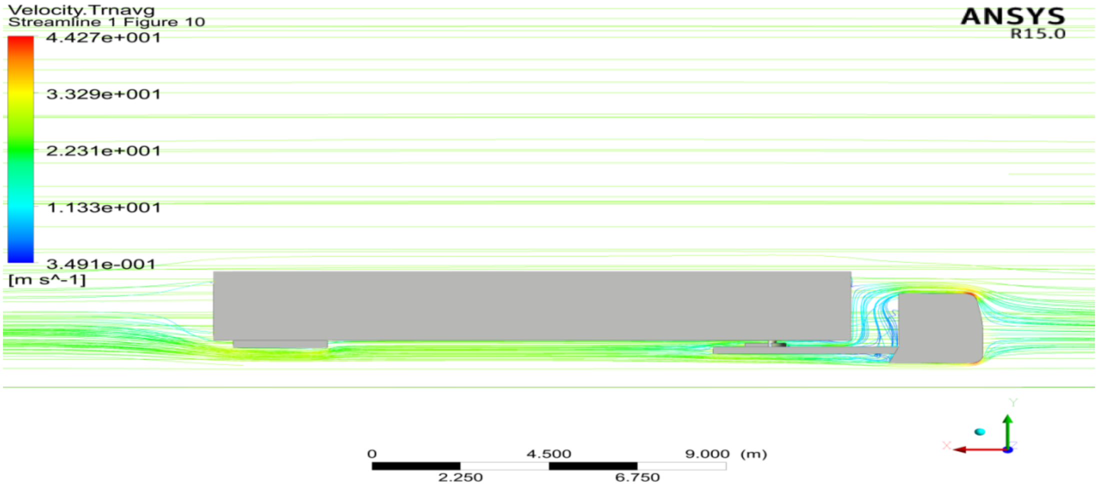
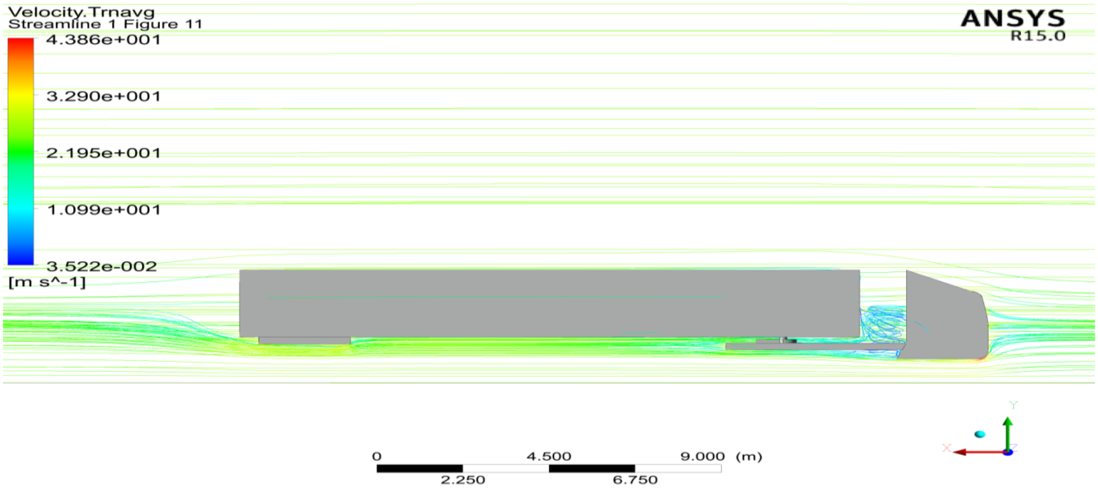


Figure 4.23. Velocity vector on XZ plane between cab and trailer for model with deflector

In the case without deflector the large amount of air is travelling to the gap from sides and top of the cab and it will effect to the regularity of the flow. On the other hand, with deflector the flow will guide directly to the top of trailer as it has been shown in Figure 4.24.



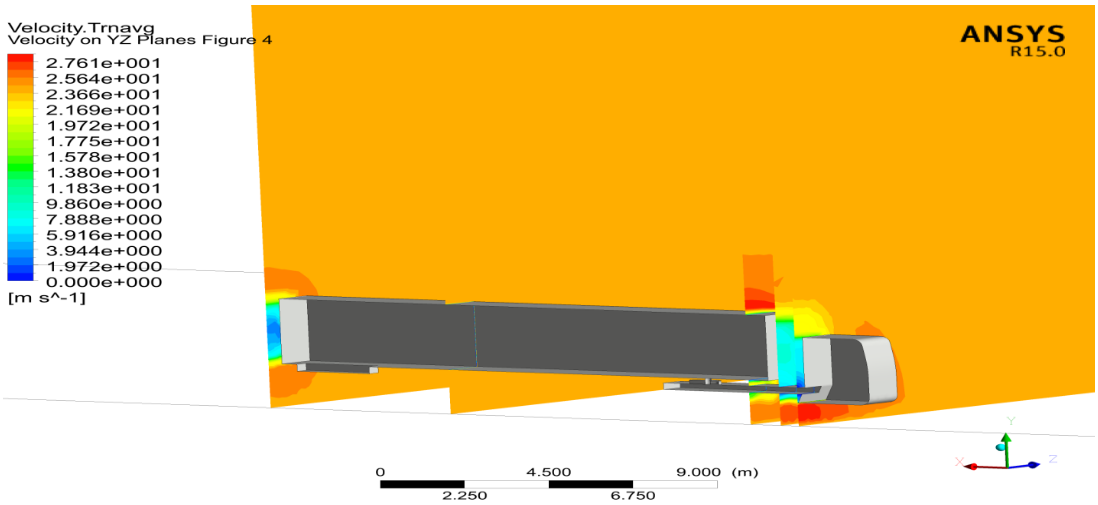
a) Without deflector



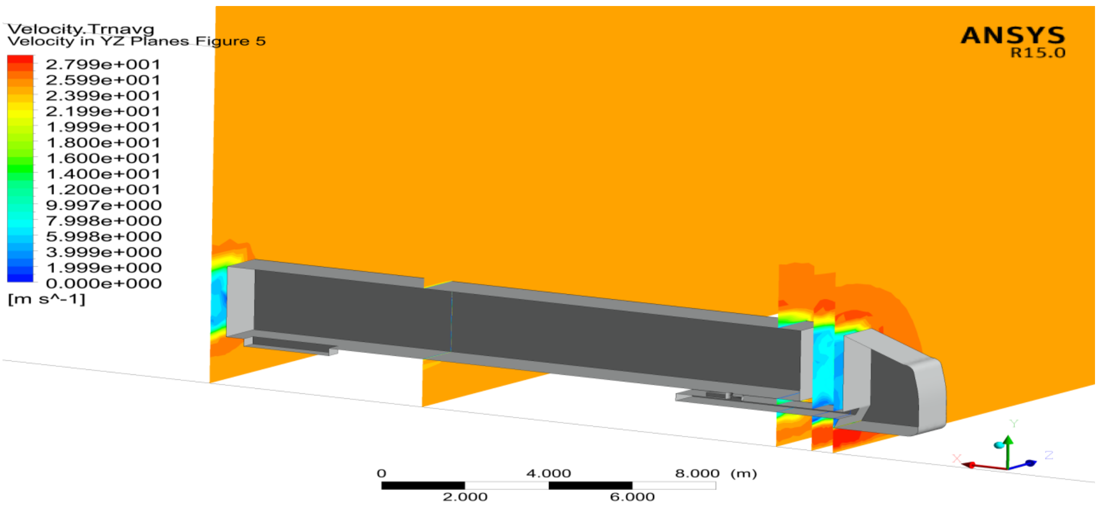
b) With deflector

Figure 4.24. Streamline released on the symmetry plane

After using deflector as it has been shown in Figure 4.25 due to the separation of the flow low velocity contains on the wakes.



a) Without deflector



b) With deflector

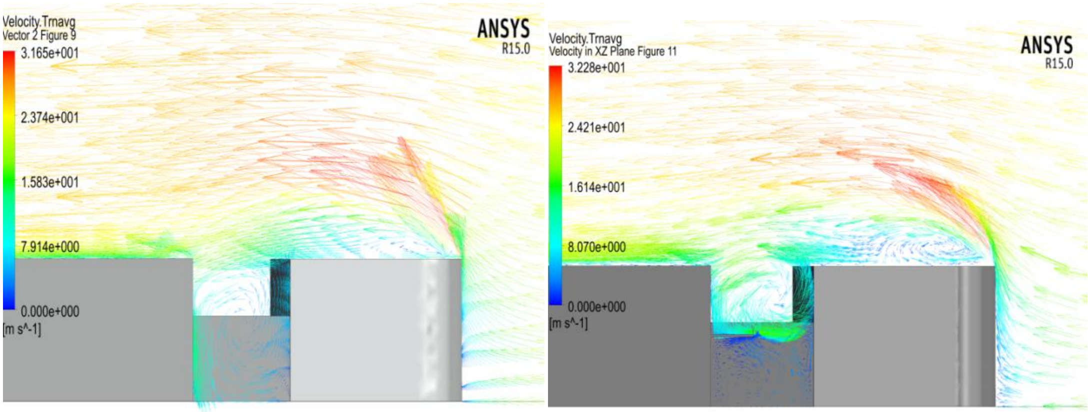
Figure 4.25.Velocity contour on the YZ planes

4.4.2 Vortex Stabilizer

As it has been shown in Table 4.3 the aerodynamic drag can reduce up to 28% with using vortex stabilizer with deflector. After using vortex stabilizer, the pressure in front of trailer will increase. This device reduces the amount of the air that has been enter to the gap to leave the gap and separate at the leeward side as it has been shown in Figure 4.26.

Table 4.3. Drag reduction with vortex stabilizer and deflector

Model	Cd	Reduction
Deflector & Vortex Stabilizer	0.68	28%



a) Without Vortex stabilizer b) With Vortex stabilizer

Figure 4.26. Velocity vector on the XZ plane in 1400 mm above ground

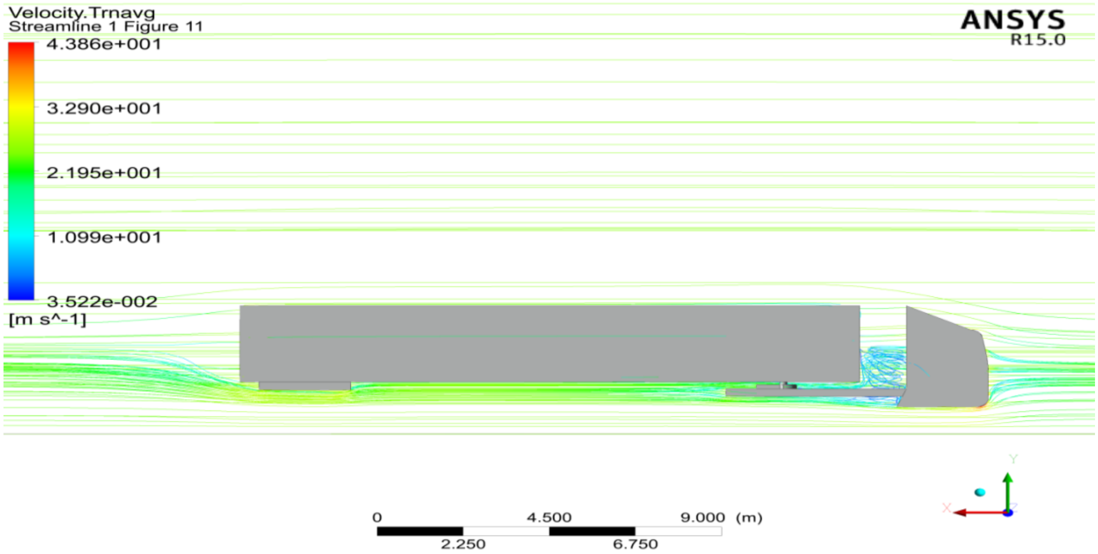
4.4.3 Teardrop

Due to installation of the teardrop on the trailer the reduction of 16% has been achieved. This device decreases the low-velocity area in the wakes. Although, the vortices will be formed more regular in the wake of trailer. On the other hand, it cannot affect a lot in the gap between cab and trailer.

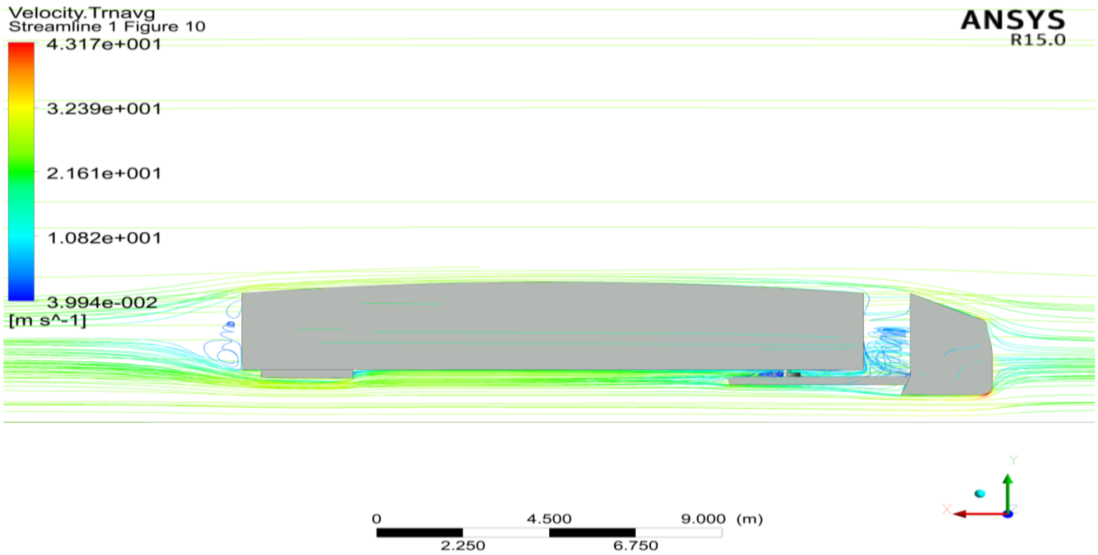
Table 4.4. Reduction of drag for teardrop

Model	Cd	Reduction
Deflector &Teardrop	0.8	16%

After creating a teardrop on the trailer, there will be extra material that have been added to the on the roof so, the frontal area of the truck is increased. However, to compare the model with the reference, the same frontal area has been used during calculation. As it has been presented in Figure 4.27 teardrop has been improved the flow structure along the roof of the trailer compare to the reference. In addition, teardrop makes the flow in the rear of the trailer downwash and the air that is exits from bottom of the trailer has a reduce in velocity, thus the up wash has been decreased. Another interesting phenomenon is that teardrop represents a well attachment over the gap when the flow has been passed from cab to trailer and on the trailer. As the flow attached to the surface from the truck and downstream of the cab, more amount of air is transferred to the gap and it can gain the disturbance of the flow in the cab region as it has been presented in Figure 4.27.



a)Without Teardrop



b) With teardrop

Figure 4.27. Streamline display in Airflow region

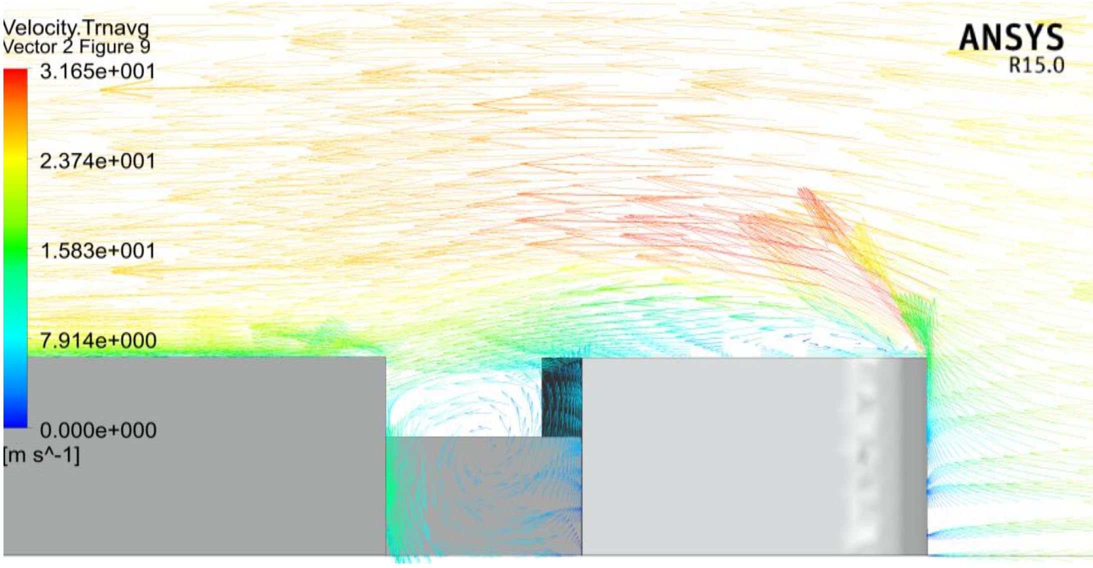
4.4.4 Gap fairing

The result for drag coefficient has been presented in Table 4.5.

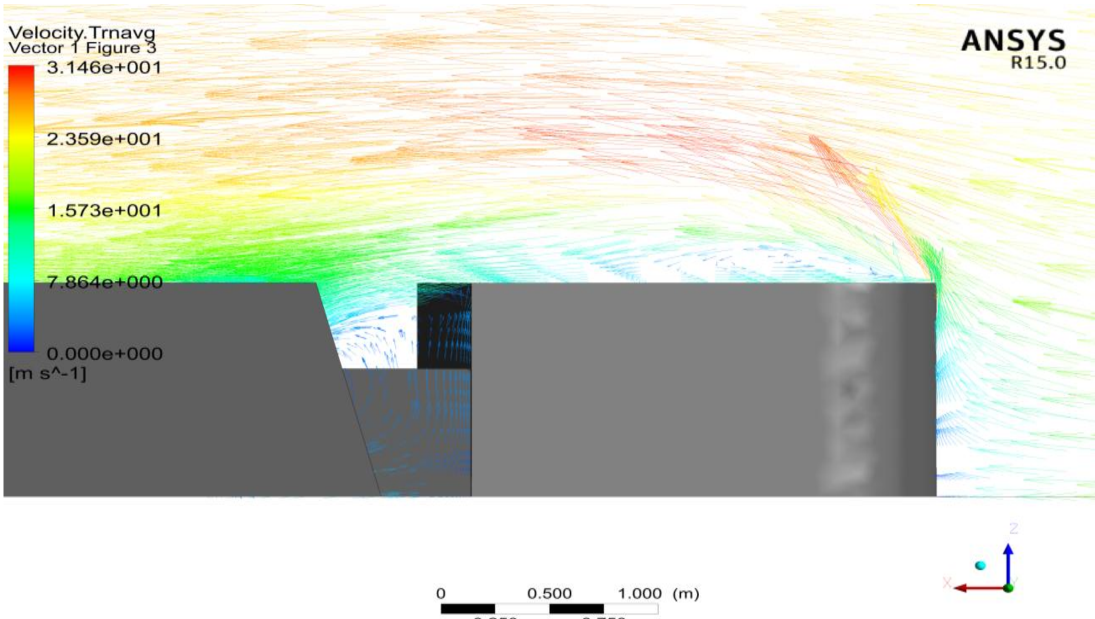
Table 4.5. Change in drag for gap fairing

Model	Cd	Reduction
Deflector & Gap fairing	0.78	18%

As it has shown in Figure 4.28 after using gap fairing the flow has been attached to the surface of trailer better than the reference model in the side of trailer. Although, air can't move on to the gap easily and when it enters to the gap it will make a vortex and small amount of the air will leave the gap.

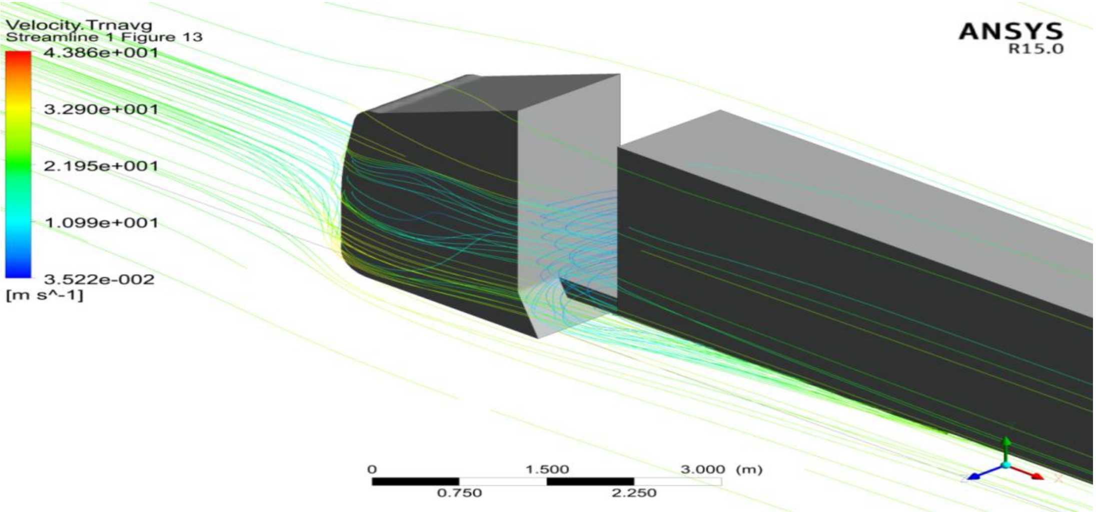


a)Without Gap Fairing

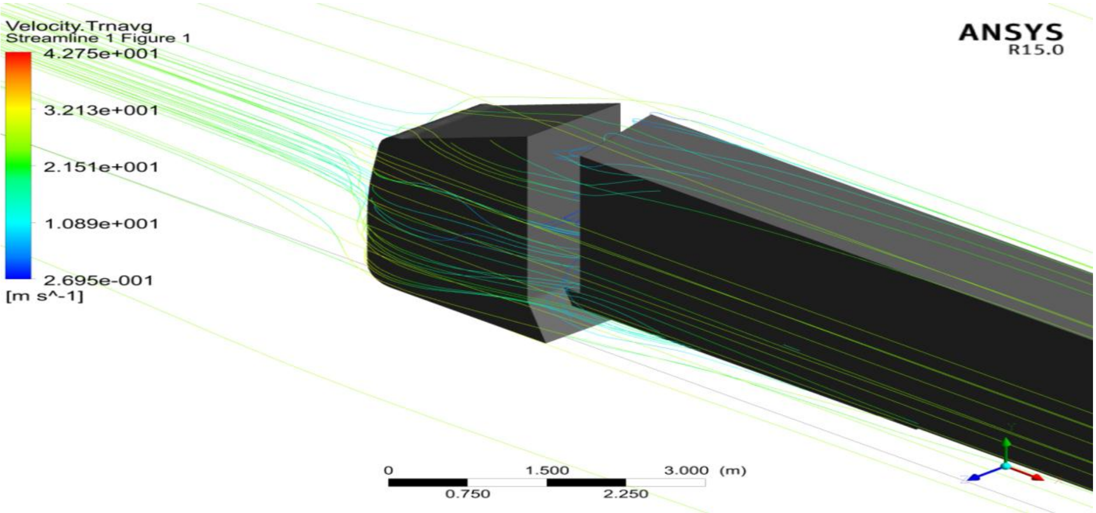


b)With Gap Fairing

Figure 4.28. Velocity Vector in XZ Plane



a)Without Gap Fairing



b)With Gap fairing

Figure 4.29 Velocity Streamline of Gap Fairing

From Figure 4.29 it is shown that when the gap fairing employed the flow can pass the gap easier and less air is going through the gap. However, the structure of the flow is more regular after using gap fairing.

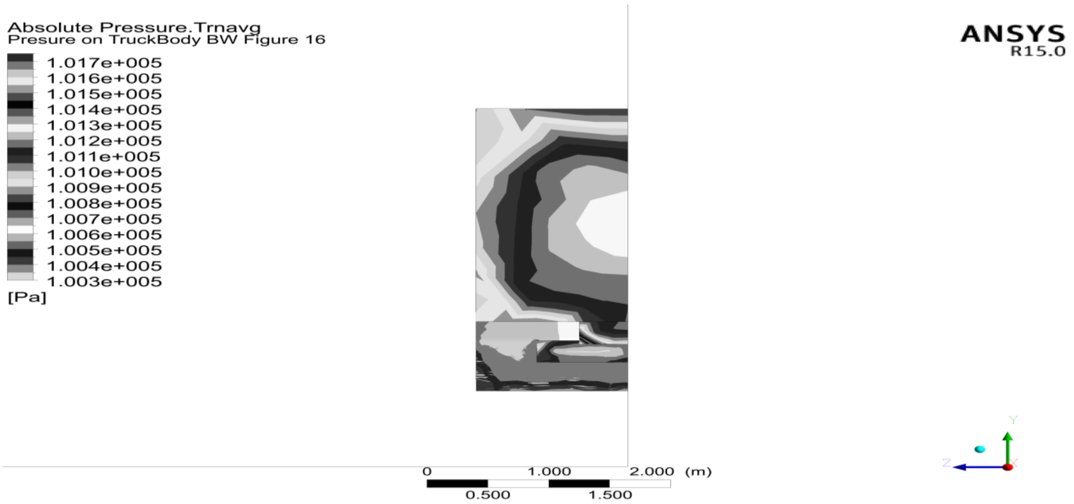
4.4.5 Frame Extension

At the end frame extension has been employed to the trailer to analyze the base treatment of the trailer and truck. Frame Extension is the combination of side plates and Base plates. The result for drag coefficient has been presented in Table 4.6.

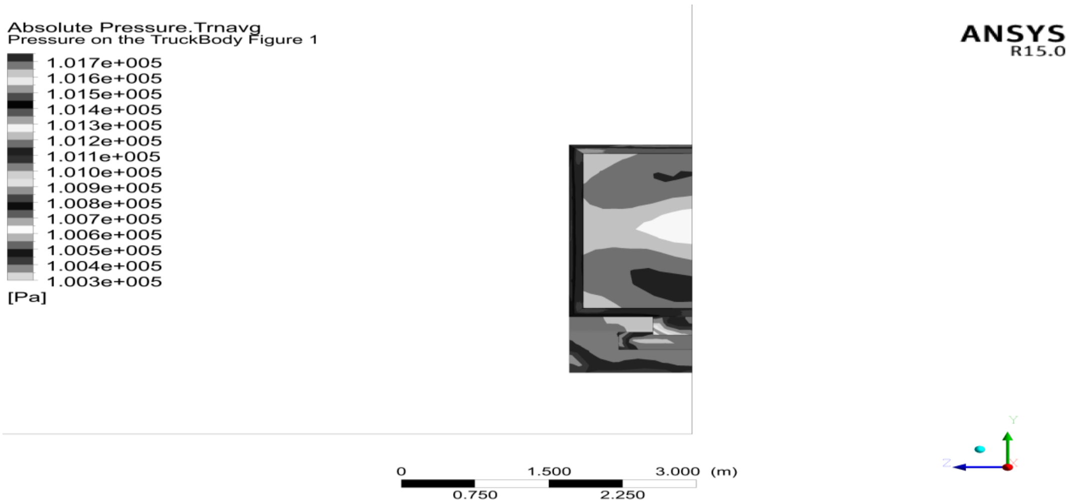
Table 4.6. Change in drag for frame Extension

Model	Cd	Reduction
Deflector & frame Extension	0.67	29%

After using Frame Extension in Figure 4.30 has been presented that the pressure distribution on the back of the trailer is almost uniform. On the other hand, due to acceleration the low pressure acting on the plates on the back of the trailer. As seen in Figure 4.15 the velocity vectors become uniform and the surface attaching of the flow has been improved in the rear of the trailer.

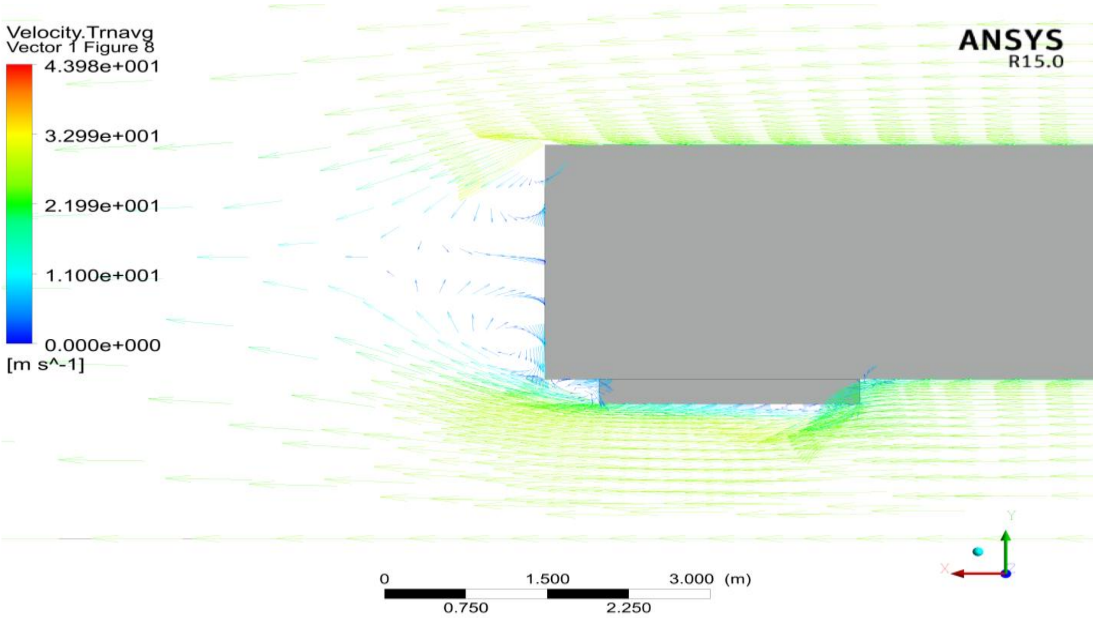


a)Without Frame Extension

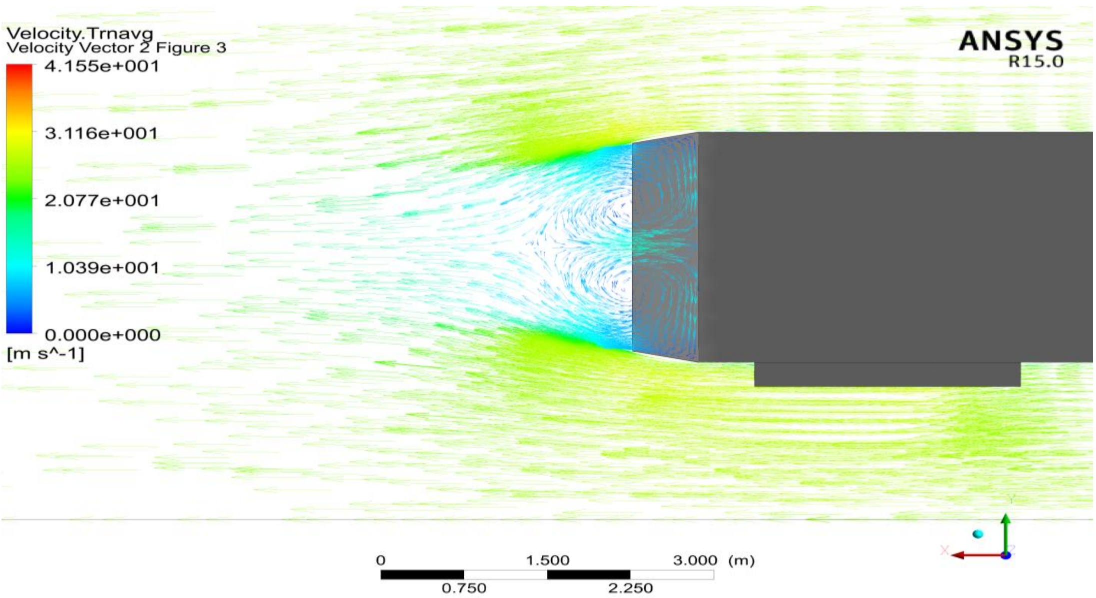


b)With Frame Extension

Figure 4.30. Pressure distributions in the rear of the trailer



a)Without Frame Extension



b)With Frame Extension

Figure 4.31. Velocity vector on the rear of the trailer

In the Frame Extension the angle of the plates has an important effect on the drag; especially the sides' plates and bottom and top plates to guide the air flow to near of the center and reduce the base wake in this condition. From Figure 4.31 it can illustrate that how the place guiding the flow to the center in the base wake to create a symmetric wake.

4.5 Key Finding

In this study it has been found that the best result for truck modifications is top fillet edge instead of sharp edge with reduction of 39% in aerodynamic drag. Furthermore, individual devices such as Deflector and Frame Extension had the best effect on drag with 42% reduction according to the simplest model (without any fillet). Combination of Deflector and Vortex Stabilizer with one percentage different had the best effect on the aerodynamic drag of truck-trailer with reduction of 41% according to the simple model. According to the analysis each device has been effect on a flow structure in specific region in the model. Deflector, Gap Fairing, and Vortex Stabilizer effect on the flow structure in the gap between truck and trailer. Deflector guide the flow on the trailer roof, Gap Fairing had better attachment on the surface of the trailer, and Vortex Stabilizer reduces the amount of flow entering to the gap. Teardrop and Frame Extension mostly effect on the flow structure on the rear of the trailer. Teardrop increase downwash and decrease upwash to make the flow structure in the rear of the trailer on the center and Frame extension increase the mean pressure distribution on the base of the trailer to decrease the aerodynamic drag.

Chapter 5

CONCLUSION

5.1 Conclusions

In this research the modification of a simplified truck and aerodynamic devices have been used to reduce the aerodynamic drag of the truck and its trailer. The fillet edges have been used instead of the sharp edges in front of the truck which causes a reduction up to 39% with top fillet edge. These modifications have been affected the flow in the gap between cab and trailer and in front of the truck. With top fillet modifications the flow can pass the cab easier and the low pressure bubbles will appear during the flow acceleration. For the cases of bottom fillet only and the Top-Bottom fillets, the results show similar pattern. However Top-Bottom fillets reduces the aerodynamic drag more effective than the bottom filet. These fillets can reduce the drag approximately 18% as it has been discussed in the previous chapter. Moreover, five aerodynamic devices such as Deflector, Vortex Stabilizer, Teardrop, Gap Fairing and Frame Extension have been employed to investigate their effects on the aerodynamic drag and wake flow structure. It has been demonstrated that combination of the deflector and the Frame Extension have been resulted in a drag reduction of 29%. In addition, deflector and Vortex Stabilizer combination also demonstrated its effectiveness by reducing the aerodynamic drag approximately 28%. The difference between these two combinations is that one of them affecting the air flow in the gap between cab and trailer and the other one affecting the flow structure on the rear of the trailer.

5.2 Future Work

Further investigations are required to investigate the effectiveness of other aerodynamic devices. It is proposed to perform additional simulation to study and compare other devices together with other parameter such as changing the angles of the plates and installation positions to find out the optimum solution to reduce the aerodynamic drag. Moreover, it is a great interest to employ different turbulent models to solve this problem to investigate suitability of the model, validity of the model and required resources for each model.

REFERENCES

Ahmed, S., et al. (1984). Some salient features of the time-averaged ground vehicle wake, *SAE Technical Paper.*

Anderson Jr, J. D. (1999). *A history of aerodynamics: and its impact on flying machines*, Cambridge University Press.

Aziz, A. A. A. & A. F. A. Gawad (2006). Aerodynamic and Heat Transfer Characteristics around Vehicles with Different Front Shapes in Driving Tunnels. *Proceedings of Eighth International Congress of Fluid Dynamics & Propulsion* (ICFDP 8).

Barbut, D. & E. M. Negrus (2011). CFD analysis for road vehicles-case study. *Incas Bulletin* 3: 15-22.

Cooper, K. R. (2004). Commercial vehicle aerodynamic drag reduction: historical perspective as a guide. *The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains*, Springer: 9-28.

Corin, R., et al. (2008). A CFD investigation into the transient aerodynamic forces on overtaking road vehicle models. *Journal of Wind Engineering and Industrial Aerodynamics* 96(8): 1390-1411.

Desai, M., et al. (2008). A comparative assessment of two experimental methods for aerodynamic performance evaluation of a car. *Journal of Scientific & Industrial Research* 67: 518-522.

Diebler, C. & M. Smith (2004). A Ground-Based Research Vehicle for Base Drag Studies at Subsonic Speeds. *The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains*, Springer: 519-531.

Duell, E. G. & A. George (1999). Experimental study of a ground vehicle body unsteady near wake, *SAE Technical Paper*.

Englar, R. J. (2001). Advanced aerodynamic devices to improve the performance, economics, handling and safety of heavy vehicles, *SAE Technical Paper*.

François, D., et al. (2009). Experimental determination of Stationary Aerodynamics loads on a double deck Bus. *11th Americas Conference on Wind Engineering*, San Juan, Puerto Rico.

Guilmineau, E. (2008). Computational study of flow around a simplified car body. *Journal of Wind Engineering and Industrial Aerodynamics* 96(6): 1207-1217.

Haruna, S., et al. (1990). An experimental analysis and estimation of aerodynamic noise using a production vehicle, *SAE Technical Paper*.

Hucho, W.-H. (1998). Aerodynamics of Road Vehicles: From Fluid Mechanics to Vehicle Engineering, *W. SAE International*, Warrendale, PA177.

Kamal, A. & M. Yamin (2006). Aerodynamic Study of a Coach Using Computational Fluid Dynamic (CFD) Technique, *Coventry University*.

Katz, J. & L. Dykstra (1992). Effect of wing/body interaction on the aerodynamics of two generic racing cars, *SAE Technical Paper*.

Lenngren, M. & C. Håkansson (2010) *CFD Analysis of Aerodynamic Trailer Devices for Drag Reduction of Heavy Duty Trucks*. Chalmers University of Technology

McCallen, R., et al. (2000). Aerodynamic drag of heavy vehicles (class 7-8): Simulation and benchmarking, *SAE Technical Paper*.

McCallen, R., et al. (2005). DOE's Effort to Reduce Truck Aerodynamic Drag Through Joint Experiments and Computations, *SAE Technical Paper*.

Miralbes, R. & L. Castejon (2012). Aerodynamic analysis of some boat tails for heavy vehicles. *International Journal of Heavy Vehicle Systems* 19(2): 115-127.

Mohamed-Kassim, Z. & A. Filippone (2010). Fuel savings on a heavy vehicle via aerodynamic drag reduction. *Transportation Research Part D: Transport and Environment* 15(5): 275-284.

Odhams, A., et al. (2010). Factors influencing the energy consumption of road freight transport. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science* 224(9): 1995-2010.

Pevitt, C., et al. (2012). A Computational Simulation of Aerodynamic Drag Reductions for Heavy Commercial Vehicles. *18th Australasian Fluid Mechanics Conference Launceston*. Australia.

Rouméas, M., et al. (2009). Drag reduction by flow separation control on a car after body. *International journal for numerical methods in fluids* 60(11): 1222-1240.

Roy, S. & P. Srinivasan (2000). External flow analysis of a truck for drag reduction, *SAE Technical Paper*.

Saunders, J. W. and R. B. Mansour (2000). On-road and wind tunnel turbulence and its measurement using a four-hole dynamic probe ahead of several cars, *SAE Technical Paper*.

Schoon, R. E. (2007). On-Road Evaluation of Devices to Reduce Heavy Truck Aerodynamic Drag, *SAE Technical Paper*.

Singh, S., et al. (2005). Effect of moving surface on the aerodynamic drag of road vehicles. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering* 219(2): 127-134.

Watkins, S., et al. (1995). Turbulence experienced by moving vehicles. Part I. Introduction and turbulence intensity. *Journal of Wind Engineering and Industrial Aerodynamics* 57(1): 1-17.

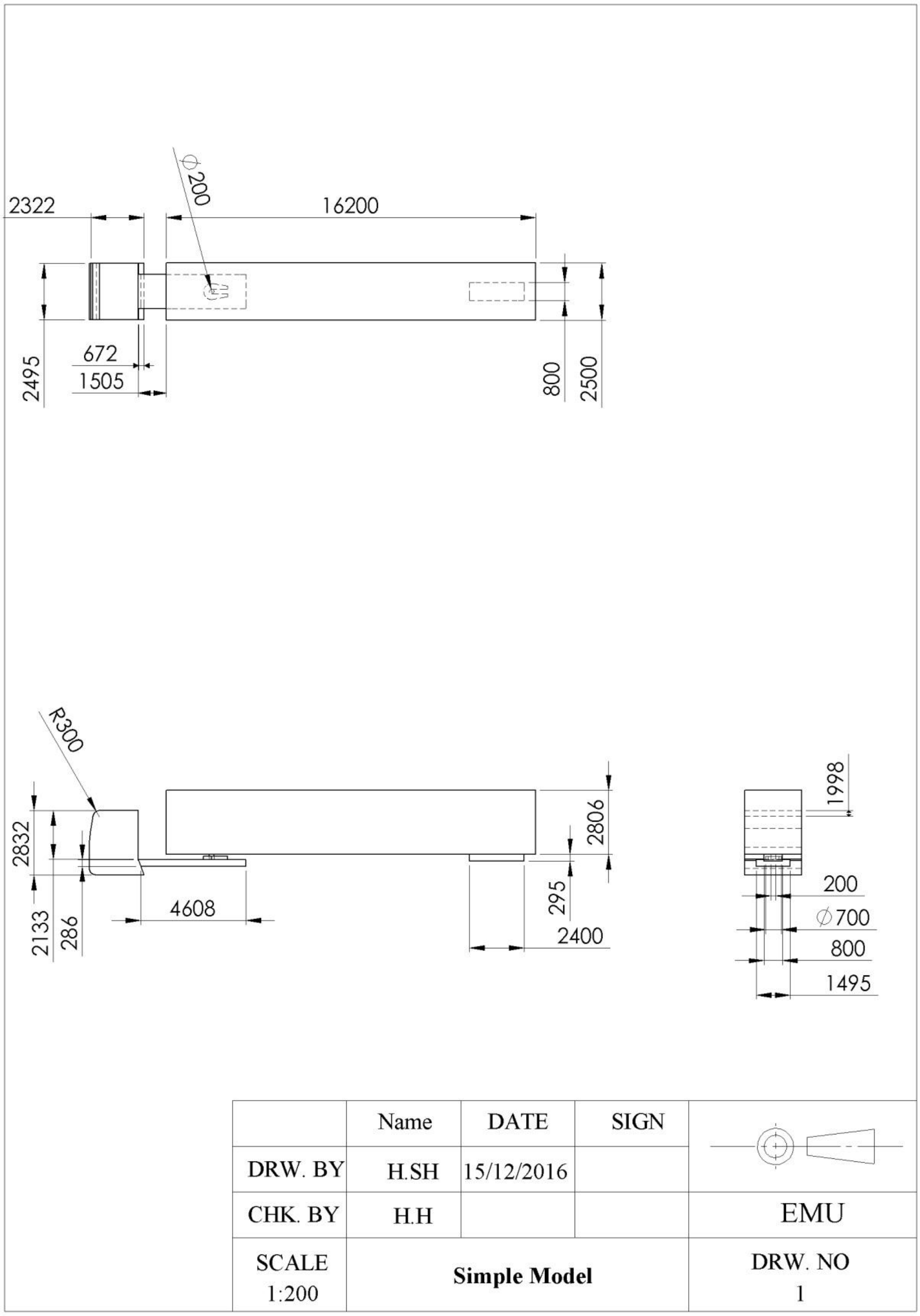
Yang, Z. & M. Schenkel (2004). Assessment of closed-wall wind tunnel blockage using CFD, *SAE Technical Paper*.

Yang, Z., et al. (2003). Corrections for the pressure gradient effect on vehicle aerodynamic drag, *SAE Technical Paper*.

APPENDICES

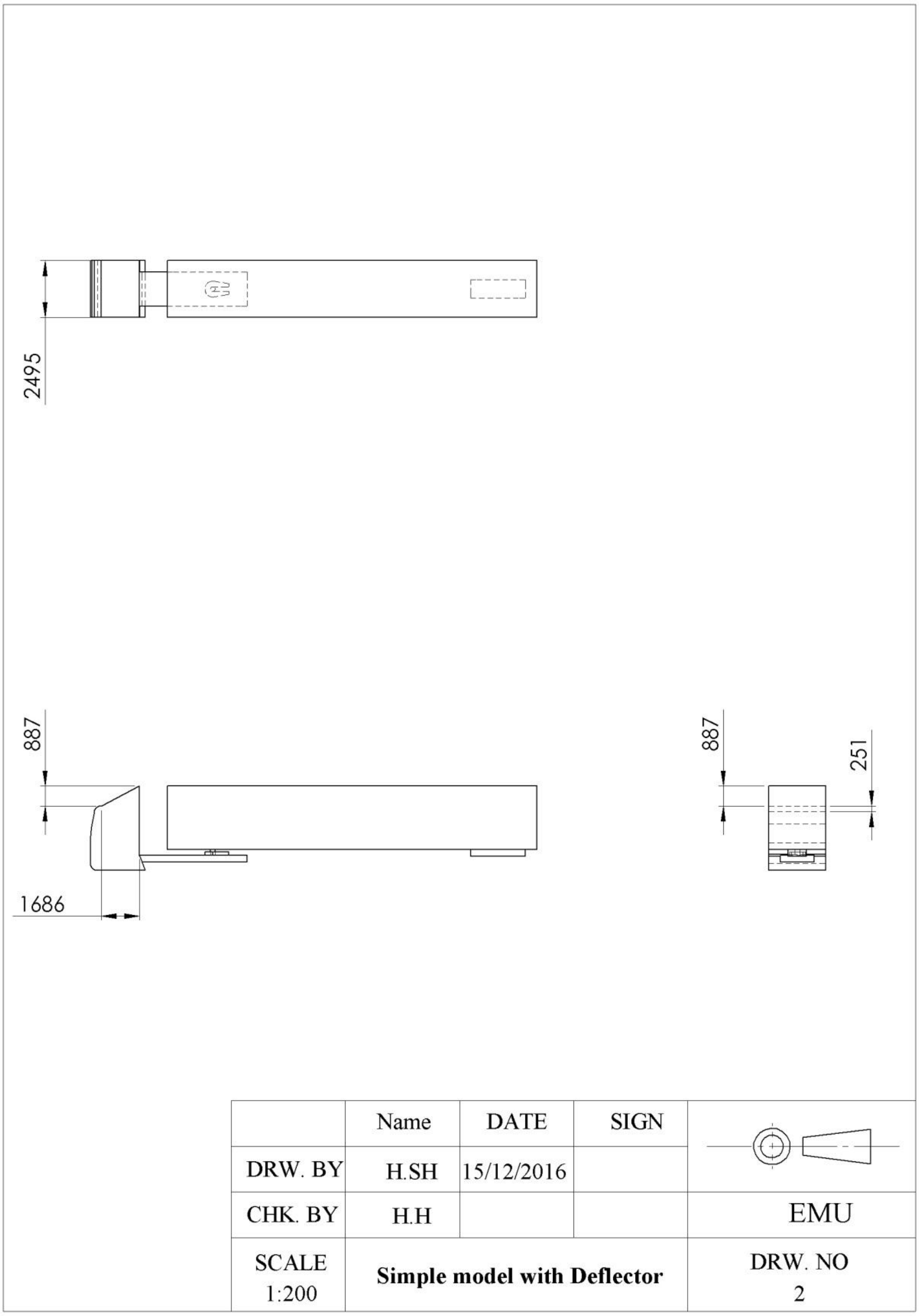
Appendix A: Technical Drawing of the Simple model Truck-Trailer

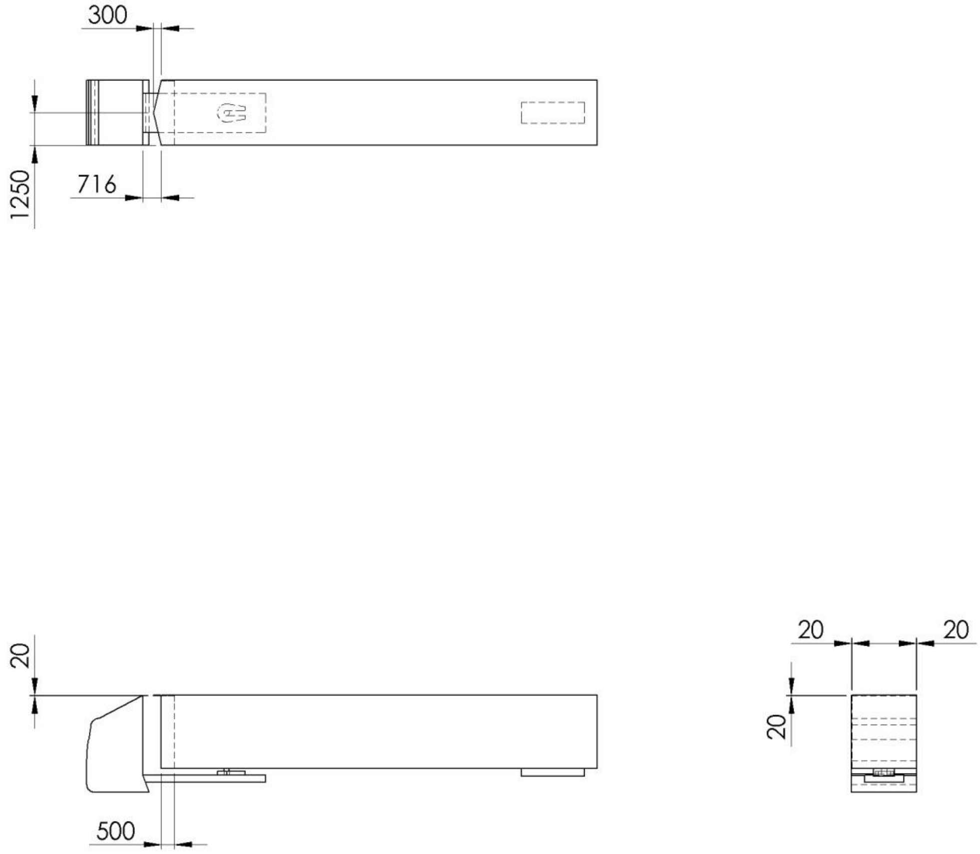
(all dimensions are in mm)



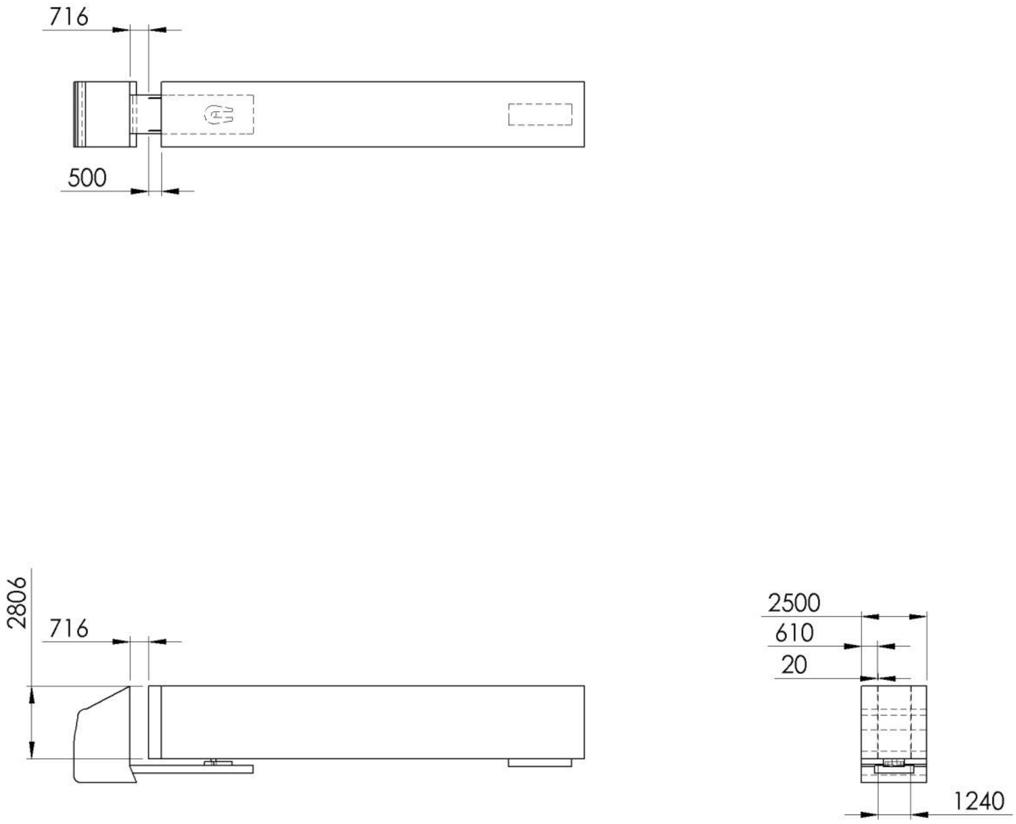
Appendix B: Dimensions of Individual Devices on Truck-trailer body

(all dimensions are in mm)

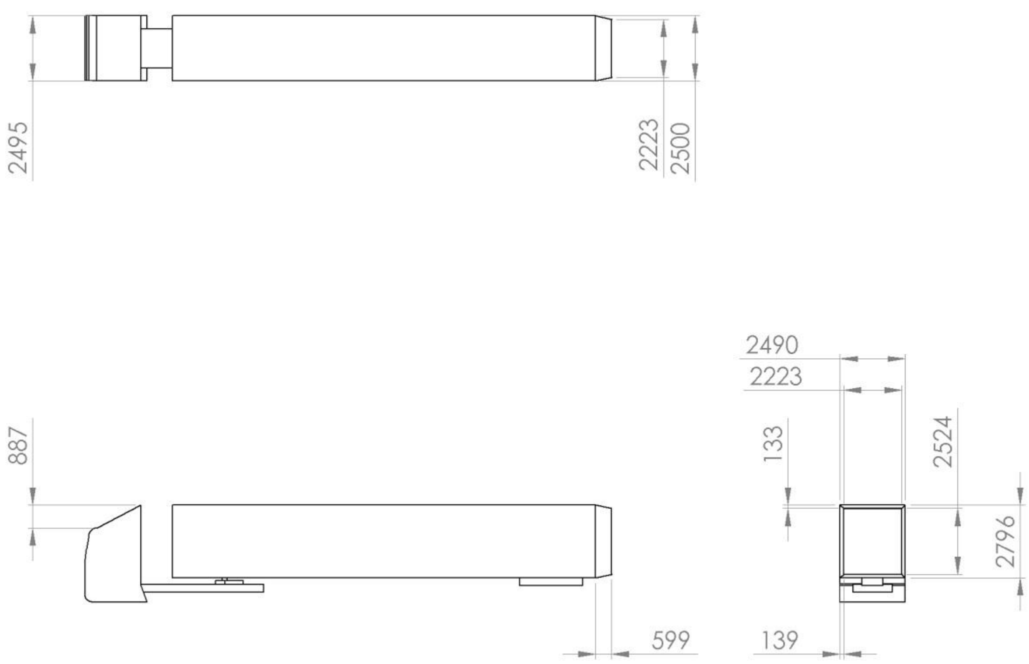




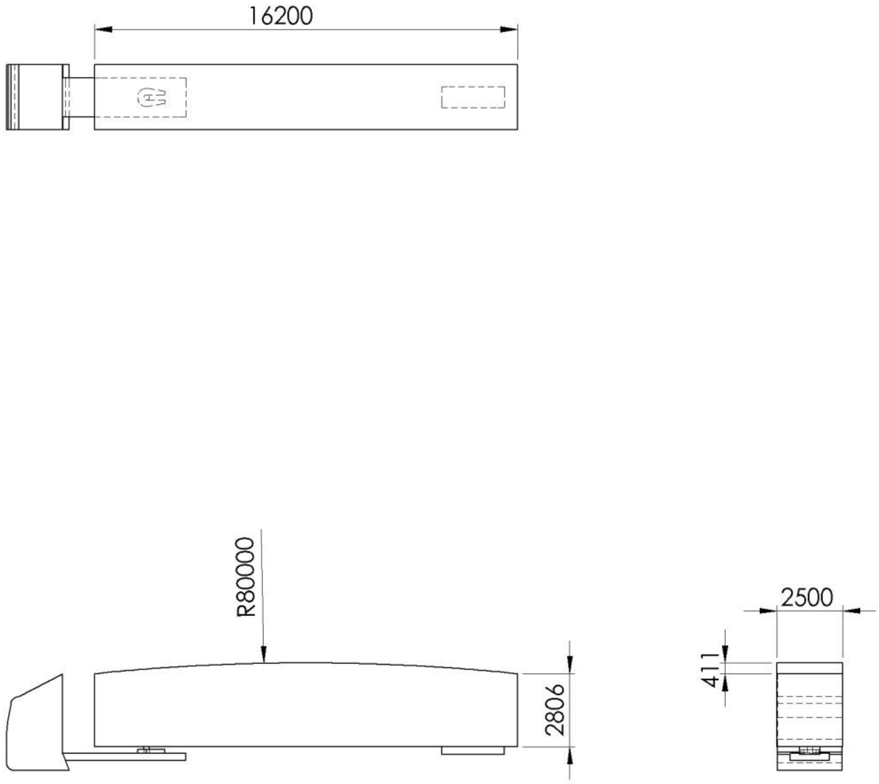
	Name	DATE	SIGN	
DRW. BY	H.SH	15/12/2016		
CHK. BY	H.H			EMU
SCALE 1:200	**Deflector and Gap Fairing**			DRW. NO 3



	Name	DATE	SIGN	
DRW. BY	H.SH	15/12/2016		
CHK. BY	H.H			EMU
SCALE 1:200	**Deflector and Vortex Stabilizer**			DRW. NO 4



	Name	DATE	SIGN	
DRW. BY	H.SH	15/12/2016		
CHK. BY	H.H			EMU
SCALE 1:200	Deflector and Frame Extension			DRW. NO 5



	Name	DATE	SIGN	
DRW. BY	H.SH	15/12/2016		
CHK. BY	H.H			EMU
SCALE 1:200	**Deflector and Teardrop**			DRW. NO 6