iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 **Issue:** IX **Month of publication:** September 2017

DOI:

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue IX, September 2017- Available at www.ijraset.com

Aerodynamic Drag Reduction of Tractor-Trailer using Wishbone Type Vortex Generators

Harshit Varshney[1], Mahish Guru[2]
[1](Department of Mechanical Engineering, JamiaMilliaIslamia, India)
[2](Department of Automotive Design Engineering, University of Petroleum and Energy Studies, India)

Abstract*: Tracter-Trailles, beingpopular commercial véhicules, are widelyused and bécasse of large frontal area and bluff bodies, an aerodynamic drag iscaused; a lot of fuel getswasted to overcomethis drag. A lowcost drag reductiondevice-Airtabdeveloped by Aeroserve Technologies Ltdwas studied for how much it reduces the turbulence caused around the tractor-trailer. This turbulence region cause's drag. Airtabs were modelled in Solidworks 2017 and then installed on a structural model of a tractor-trailer. Vortices caused by Airtabs were studied in a steady state flow simulation on Fluent of Ansys 18.1 workbench. The paper gives specific guidelines to get close to accurate results in the simulation using the correct meshing criteria, appropriate model chosen for such type of flow cases, boundary conditions applied to the domain, application of wall functions in different turbulence models, right solver settings in Fluent to get easy convergence, suitable place for installation of Airtabs and monitoring the value of drag coefficient. Comparison of contours of velocity and turbulent kinetic energy is also done to show the reduction in turbulent (wake) region. Mesh independence study and validation of results prove reliability of results which help in increasing the fuel economy of tractor-trailer.
***Keywords*:Aerodynamic Drag; Airtabs; Fluent; K-Epsilon; Tractor-Trailer; Vortex Generator; Wake Region.**

I. INTRODUCTION

Increase in fuel pricing (diesel and petrol) and continuous depletion of these resources have led to many researchers, engineers and scientists coming up with ideas to get more fuel economy. Reducing drag is an aerodynamic practice that is very common and is done by reducing frontal impact area in high end sports cars. Trolleys and trailer-trucks (tractor-trailers) being commercial vehicles used for transportation of heavy goods, take up a lot of fuel, so fuel economy is an important factor here as due to large frontal impact areas of these vehicles, drag acting on these bodies is quite considerable. Many devices are used to reduce this aerodynamic drag, one of which is a vortex generator.

II. RELATED WORK

Many researches have been done on the aerodynamic study of tractor-trailer [15] and also on drag reduction in sedan and hatchback cars by using different vortex generator models. [4]

At a constant speed of 90 Kmph, 50 percent of the total power sent to the wheel is used up in overcoming the aerodynamic drag [1]. Airtabs (a commercial type vortex generator) generally save up to 2.3 percent fuel consumption of the tractor-trailer by reducing drag by up to 1.1 percent [2]. Many trailer-truck experimentations using the other aerodynamic devices were done by wind tunnel testing and the data gathered is in the following table [3].

Table I
aerodynamic drag reducing devices

	Change in C_D	Annual Fuel (L)
Base Drag Reduction		
Aerovolution inflatable trailer rear fairing	0.0438	2638
Trailer vortex strakes -	0.0195	-1174
Trailer leading edge fairings		
Freight Wing Inc. NXT Leading Edge Fairing w/o roof fairing	0.0369	2222
Manac prototype trailer leading edge fairing	0.0335	2017
Underbody Drag Reduction		

Freight Wing Belly Fairing (low rider)	0.0478	2879
Francis Cardolle trailer wheel fairings	0.0078	470
Gap Sealing		
Volvo cab-extender extensions	0.0123	741
Labyrinthine tractor-trailer gap seal	0.0018	108

III.PRE- REQUISITES

Aerodynamic drag is a force that is exerted on a body moving in air. This force acts opposite to the direction of motion. Major types of drag acting upon a body in motion in air are: friction drag and pressure drag.
Pressure drag is caused due to difference in pressure between opposite sides of a body, whereas the friction drag exists due to shear stress between body of the object and the fluid.

A. Wake region

For a fluid particle flowing around a cylinder, the pressure is maximum at the stagnation point and it gradually decreases along the front half of cylinder. This region is favorable for flow and the flow remains attached. However, in the rear half of the cylinder, the pressure starts to increase. As a result, the flow starts separating from the surface and this creates a highly turbulent region behind the cylinder. This region is termed as wake region. The pressure inside the wake region is low as the flow separates and a net pressure drag is produced [5].

B. Pressure Drag in Tractor-Trailer

A tractor trailer can also be taken as a body with multiple curvature points where the flow separation might exist, resulting in a turbulent wake region which would create a pressure drag; therefore decreasing the fuel economy.

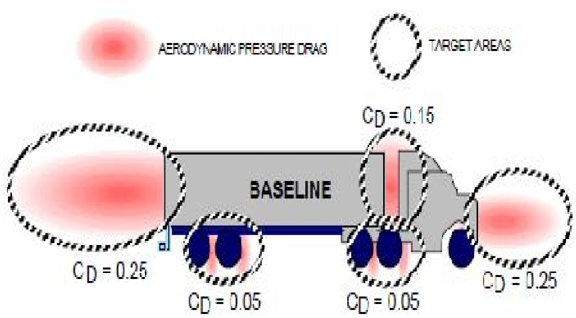


Fig.1 Pressuredrag distribution

As can be seen from the above representation, there are three major regions where aerodynamic pressure drag exists. However, at two places, i.e. at the rear end of the trailer and the region between the tractor and trailer, the use of vortex generators can reduce the wake region and thus reduce the aerodynamic pressure drag.

C. Vortex generators

Vortex generator are the devices that help in reducing the wake region by delaying the flow separation point, which they achieve by generating vortices. Vortex generator generally energizes the boundary layer flow by creating stream wise vortices which mix the free stream and pull this free stream flow back into the velocity boundary layer. As a result of this mixing of high energy free stream flow with the lower energy velocity boundary layer, the energy of the boundary layer increases, thus decreasing the effect of the adverse pressure gradient and hence the flow separation is delayed. Due to this the wake region narrows and the pressure drag decreases.
As it can be seen that the percentage improvement is maximum in Airtabs (which are a wishbone type vortex generator) [2], hence they will be used for our study.

TABLE II
Drag reduction comparison

	Drag Force (N)	Drag Coefficient	% Improvement
No VG's	838.385	0.861	-
Airtabs	829.054	0.851	1.11
Vortex Strake Device	840.549	0.863	-0.26
Delta Wing	830.578	0.853	0.93

IV. EXPERIMENTAL RESULTS

A. Airtab design

It is a wishbone type vortex generator which has been patented by Aeroserve Technologies Ltd [6]. Design of Airtabs is inspired by the bio-mimicry of the finlets of the fastest tuna fishes, therefore this device is also called eco-fins.

These Airtab geometry parameters were designed using iterative processes and the design was somewhat inspired from the Vane type vortex generator. These Devices extend out of the boundary layer without any sharp edges therefore the height of this type of VGs is set for various boundary layers generated with respect to the speed and span of the trucks. The dimensions of the Airtab are given in millimeter (mm).

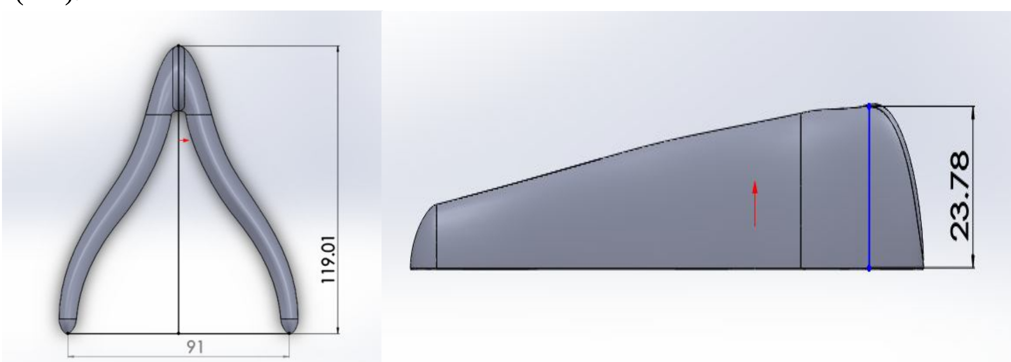
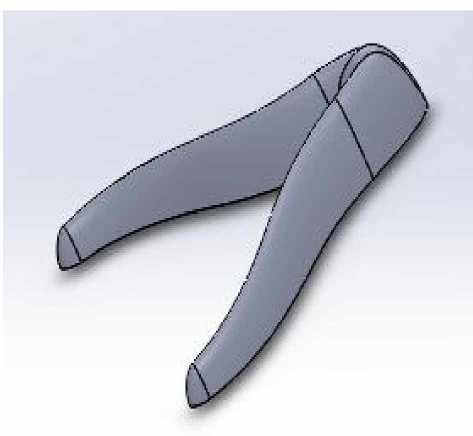


Fig. 2Dimension of Airtab (in millimetre)



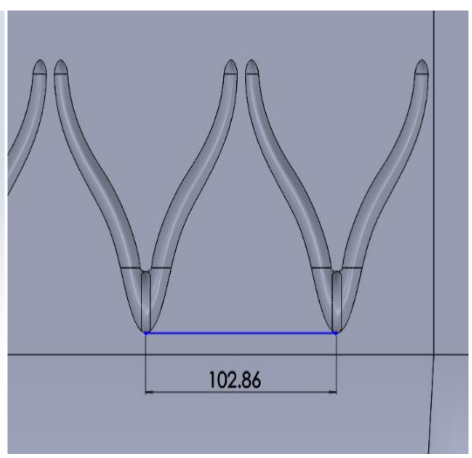


Fig. 3Solidworks model of AirtabFig.4 Spacing between the Airtabs

B. Modelling

An Aerodynamic CAD model of the tractor-trailer having the following dimensions was made on SOLIDWORKS 2017 after referring to the guidelines given for real life tractor-trailers. This was later imported into ANSYS FLUENT.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue IX, September 2017- Available at www.ijraset.com

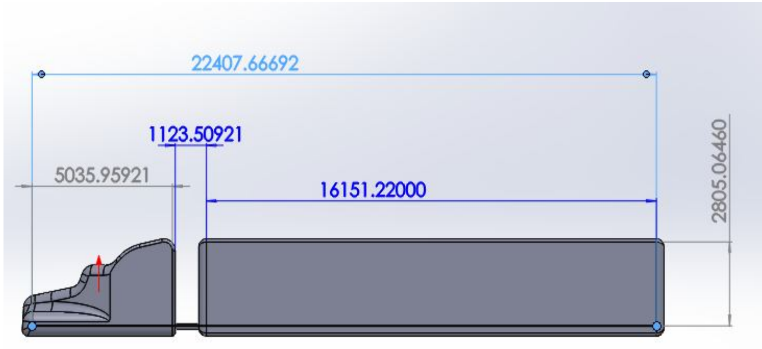
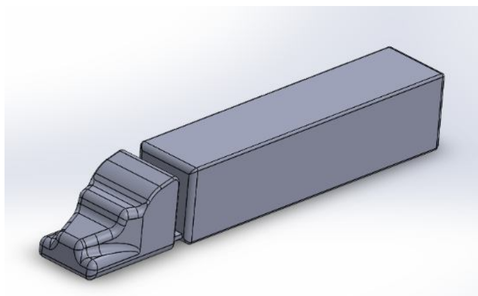


Fig. 5 Side view



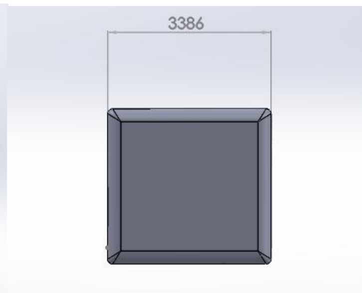


Fig. 6 Isometric view

Fig. 7Rear view

C. Pre Processor

1) *Enclosure and Boolean geometry:* Ansys 18.1 workbench was used to access Fluent. The tractor-trailer model was imported into design modeller to create a domain of the airflow that encompasses the body. This is done usually to have a virtual CFD application of the wind tunnel testing. Generation of this domain was done by using the Enclosure feature. Tractor trailer, being large in dimensions, would take a lot of computation time. So, to cut down the computation time and the CPU usage the body was cut in half using the symmetry feature of Enclosure. Tractor trailer, being a symmetric body, would give us the same result. The Drag force calculated for the body by this experiment will be half the drag force actually acting on the body.

The enclosure generated was of the dimensions such that it was approximately two lengths forwards from the body and three lengths backwards, four times the height upwards and tire length (900 mm roughly) downwards. Then Boolean feature of design modeller was chosen to subtract the body from the domain. Named selections are very important to give to the enclosure because these decide the boundary conditions applied in Fluent afterwards.

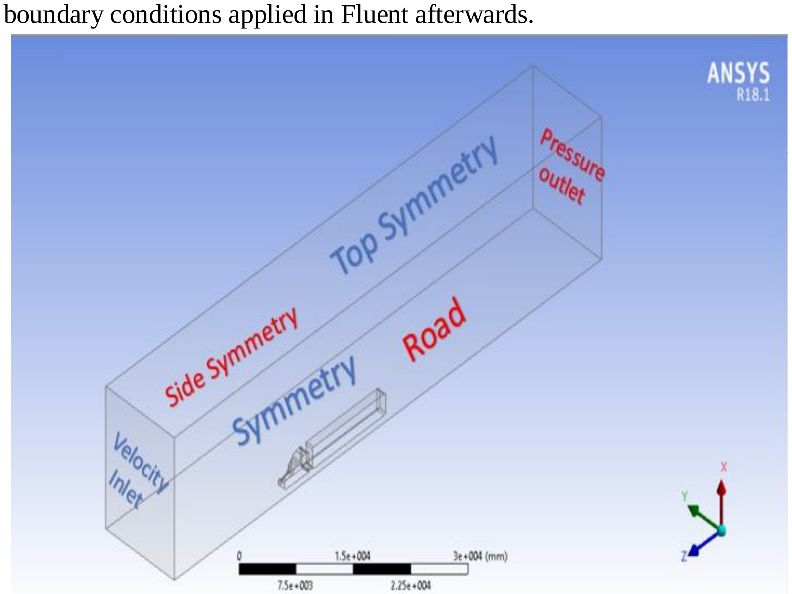


Fig. 8Enclosure

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue IX, September 2017- Available at www.ijraset.com

2) *Meshing:* Meshing is a very important aspect of the aerodynamic steady state analysis in CFD. A lot of parameters loosely or completely depend on the quality of the mesh. While doing the meshing, sizing features were changed to proximity and curvature, relevance centre was tuned to fine, maximum size of the element was given to be 800mm and minimum size was set at 5mm. Also, the triangle surface mesher was set to program controlled.

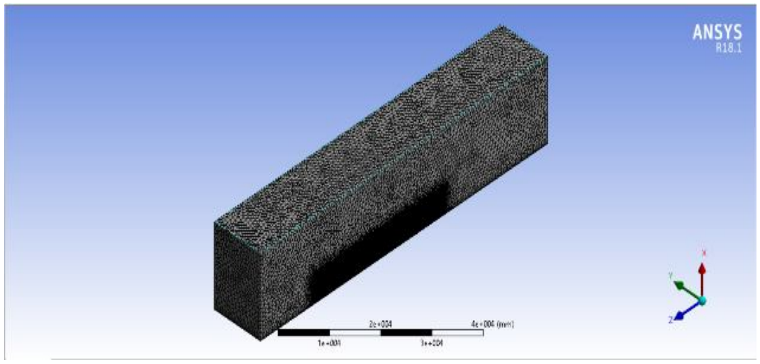


Fig. 9Mesh Isometric View

3) *Body sizing:* From the previously illustrated diagram of tractor-trailer, we have the knowledge that the turbulence created around the body is largely responsible for the pressure drag. In order to accurately capture the flow around the body and the vortices reducing this drag and how turbulence is being formed and how it amounts to pressure drag, we have to make a very fine mesh around the body. So, a very fine mesh was locally made around the body using body sizing features having minimum element size as 75mm.

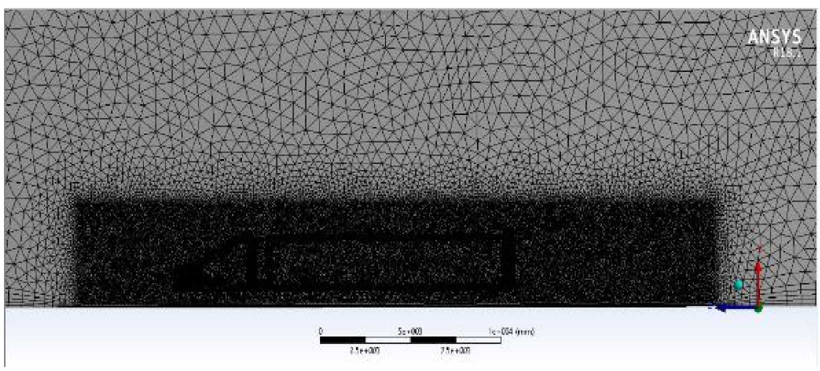


Fig. 10Body sizing

4) *Y plus /Inflation layer*

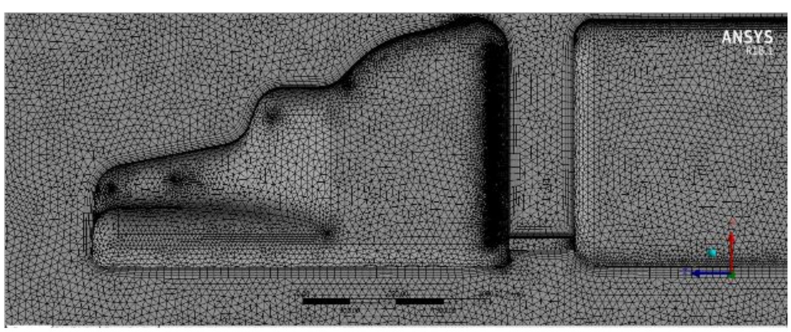
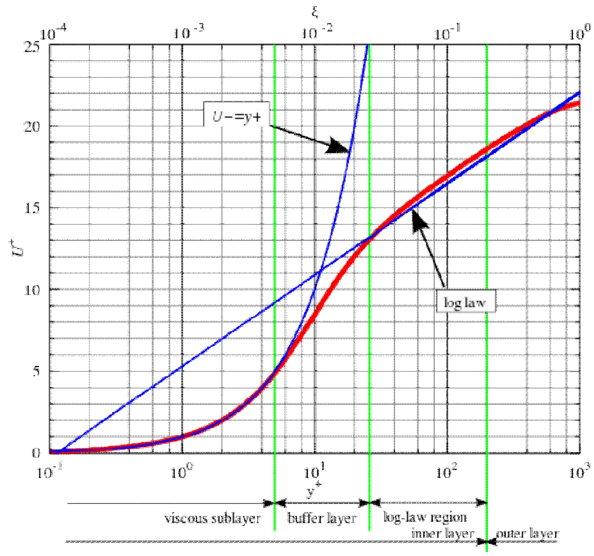


Fig. 11Inflation Layer

Another important feature for drag production is the boundary layer separation, and also for proper functioning of wall function, Y-plus value is implied to the model using inflation layer. Y-plus value is basically used for resolving the viscous sublayers. Y-plus = 0 at the wall adjacent grid, First a proper value of the Y-plus was determined by using the graph for the turbulence model and the wall treatment that we will be using.



Graph 1 – Log-Law Y+ region

We will be using the K-Epsilon Realizable model with Non-equilibrium wall function. The boundary layer formation at the end of the trailer becomes turbulent because of excessive flow over flat surface and thus the Y-plus value of the mesh around the boundary should lie between 30-300(in *the* Launder *and* Spalding'slog law inner region) [7] By trial and error of the following formulas, the first layer thickness (Y_1) was determined and inflation layers were generated.

$$Re = \frac{\rho \cdot U \cdot L}{\mu}$$

$$y+ = \frac{rho \cdot U_\tau \cdot \Delta y_1}{\mu} \quad \rightarrow \Delta y_1 = \frac{y + \cdot \mu}{\rho \cdot U_\tau}$$

Here, L is the length of the body, U is free stream velocity, ρ and μ are the fluid density and viscosity respectively and y_1 is the value of first layer thickness [13].

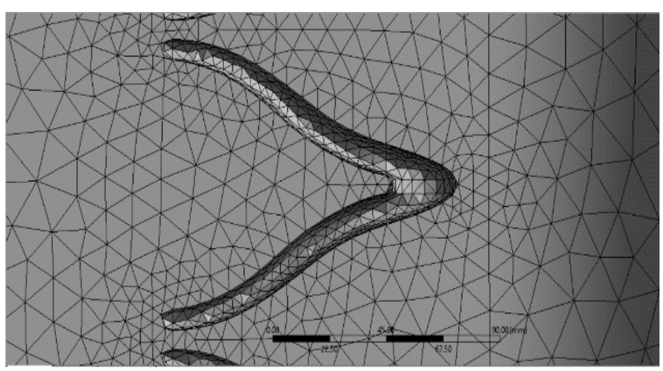


Fig. 12 Fine mesh

The mesh had Nodes-1679098, Elements – 8889066 without Airtabs and Nodes-2079544, Elements – 10320354 with Airtabs

V. C.F.D. MATHEMATICAL SOLVER

A. Turbulence Model

Turbulence model chosen for this simulation is K-Epsilon Realizable for doing steady state analysis of tractor-trailer. It is the most prescribed model for turbulent simulations in commercial packages as it performs better for flow separations, adverse pressure gradients, rotations and recirculation and also for bluff bodies[8]. This model is development in the deficiencies of the Standard K-Epsilon which is a two equation model as it contains the formulation for turbulent viscosity using eddy viscosity formula and it has transport equation for mean square vorticity fluctuations for dissipation rate[9].

RNG and Realizable both show considerable improvements over the standard K–epsilon where the flow features include strong streamline curvature, vortices, and rotation. However, early researches have shown Realizable showing better and fast converged results for several validations of separated flows [9].Optimization of results in reduction of drag was tested in both RNG and Realizable K-Epsilon model. RNG model showed a lot of waves and instability in residuals and monitors (very bad convergence) whereas Realizable model took more time for computing but converged in lesser iterations and gave accurate drag reduction result.

B. Wall functions

Out of enhanced and non-equilibrium wall function, the latter was chosen because it works better in adverse pressure gradients. It is a two layer based model and works on Launder and Spalding's log-law for mean velocity and is sensitized to pressure-gradient effects. Being in the log-law region, the Y-plus value should lie between 30-300. But for the enhanced wall treatment Y-plus = 1 at wall adjacent faces and to achieve that, the size of the mesh element near wall should be very small. This in turn increases the computation time.

C. Boundary Conditions and Reference Values

For simulation, the boundary values which need to be applied are as follows:

1) Velocity inlet: Free stream velocity is equal to vehicle velocity (95 Kmph) = 26.389 m/s, Turbulence intensity is kept 1 % because almost no turbulence exists at inlet.
2) Bottom surface of the enclosure (i.e. Road) has wall as boundary condition which means wall shear occurs here. It has a no-slip condition.
3) Pressure outlet: Outlet is given a gauge pressure or 0 i.e. 1 atmospheric pressure (101325 pascal) which means the domain is at atmospheric pressure. Turbulence intensity is 5% here.
4) All other faces are given boundary condition as symmetry. Symmetry boundary condition does not necessarily means that there is mirroring of some kind, it just means these walls are having no wall shear (or do not have no-slip condition)

Frontal area was calculated using the report- projected area feature of Fluent. Reference values were then set to calculate the drag force and coefficient of drag. Reference values were computed from inlet. Drag force and coefficient monitors were set under report definitions. These values are as follows

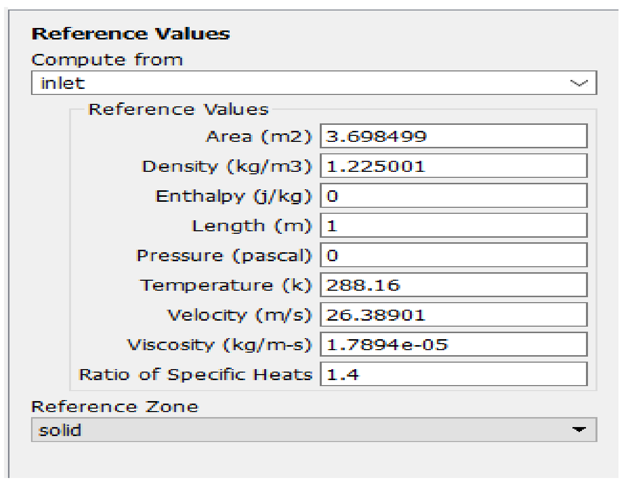


Fig. 13Reference values

D. Solver

If a SIMPLE solver was used for pressure-velocity coupling (which is very important for our Turbulence – drag relation) it would take around 3000-4000 iterations to converge the result. But instead of that, Couple Solver was used with which the convergence takes at a maximum 1000 iterations only. But the couple solver does take more memory and CPU usage. It allows the pressure-velocity based algorithm to solve in a segregated way which obtains an efficient and robust single phase implementation of steady-state flows with superior performance to segregated scheme. While using this Coupled Scheme one needs to change some values with it. These values are set as: pressure relaxation factor = 0.25, momentum relaxation factor = 0.25, Turbulent viscosity = 0.8. The gradient of solution variable at the cell centre was set to Green Gauss node based because it is more accurate, minimizes false

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue IX, September 2017- Available at www.ijraset.com

diffusion and is recommended for triangular meshes. The turbulent viscosity ratio was changed to maximum value of a 100 million because in our simulation high turbulence regions were generated and Fluent limits the turbulence ratio in certain cells which hampers the result and disrupts the convergence [12].

E. Solution

The Tractor-trailer, being large in dimensions, and having a refined mesh would take a lot of time to iterate. To minimize the time taken for convergence, the solution was initially run on First order momentum, First order turbulent kinetic energy, standard cell-face pressure and First order turbulent dissipation rate with turbulence viscosity 0.8 to achieve convergence early in 300-400 iterations. After the solution is converged it is run for 800 more iterations (until converged) on Second order momentum, Second order turbulent kinetic energy, PRESTO cell-face pressure and Second order turbulent dissipation rate with turbulence viscosity 0.95 to get a more accurate value of drag force/coefficient as it uses large stencil which are very essential for triangular or tetragonal mesh and PRESTO is used for highly swirling flows using steep pressure gradient [12].

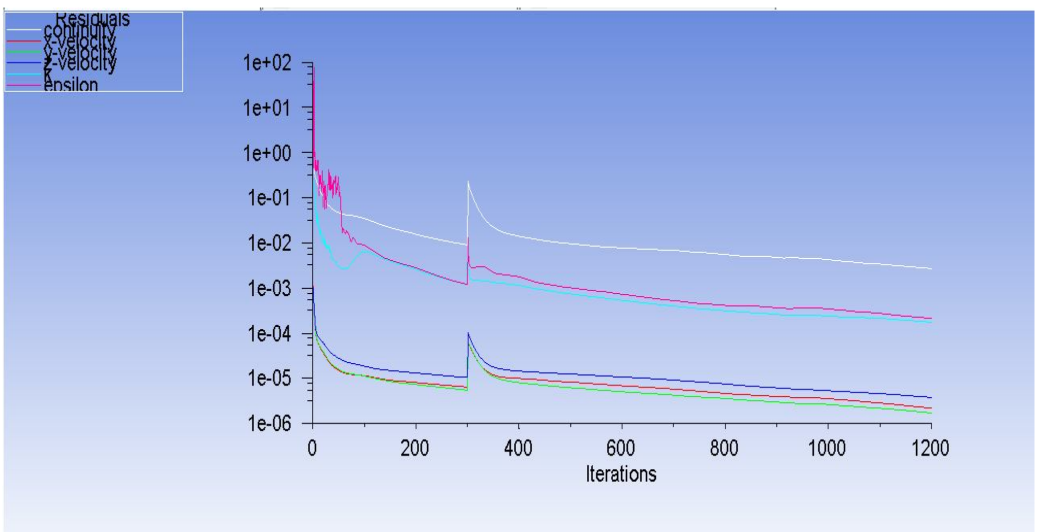
Drag coefficient calculated by Fluent is for the force acting on half of the body, so total force is double this force. The drag coefficient is calculated using the formula given below. F_D is drag force, A is frontal area, Rho is air density and $V_o$is free stream velocity.

$$C_D = \frac{F_D}{A\left(\frac{\rho V_0^2}{2}\right)}$$

VI. EXPERIMENTAL SIMULATION AND ANALYSIS

A. Before vortex generators

Drag forces were monitored as follows

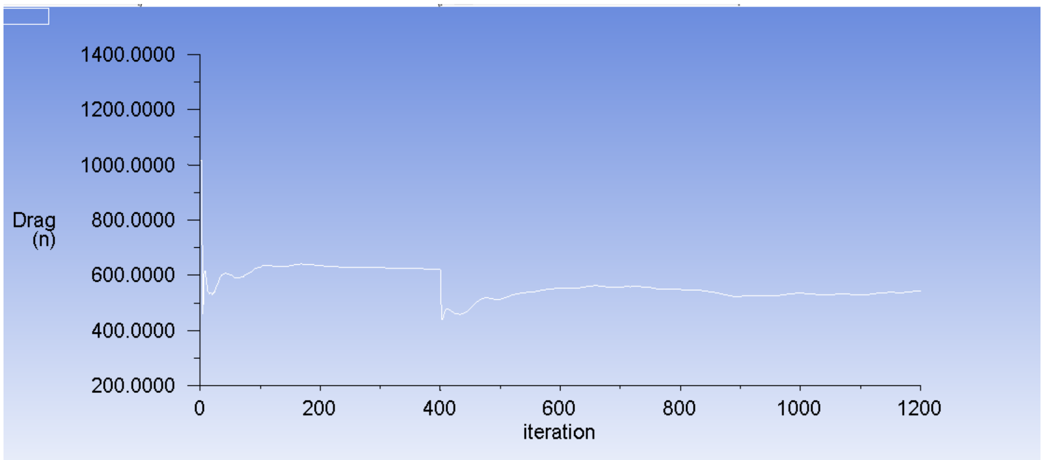


Graph 2 – Residual Curves (without Airtabs)

Forces - Direction Vector (0 0 -1)

	Forces (n)		
Zone	Pressure	Viscous	Total
carbody	444.1189	100.47333	544.59222
Net	444.1189	100.47333	544.59222

Fig.14 Force report (without Airtabs)



Graph 3 – DragMonitor (without Airtabs)

The Velocity and Turbulence Kinetic Energy contours are shown below when solution was obtained without installing vortex generators.

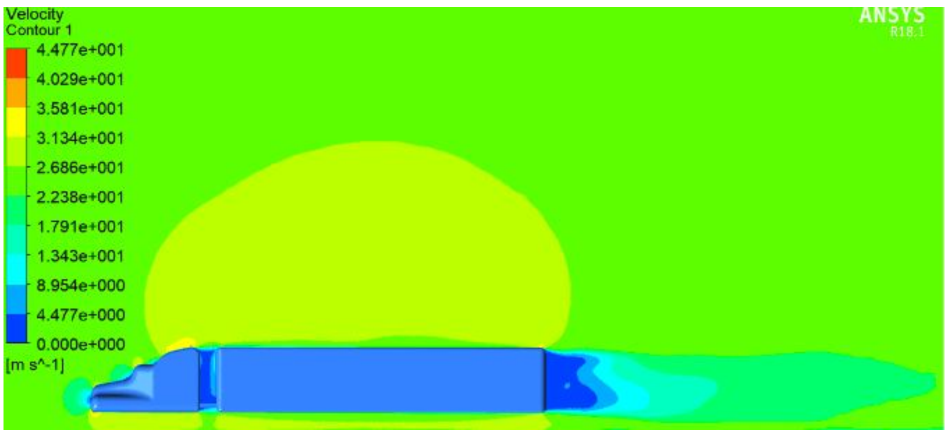


Fig.15Velocity Contour (without Airtabs)

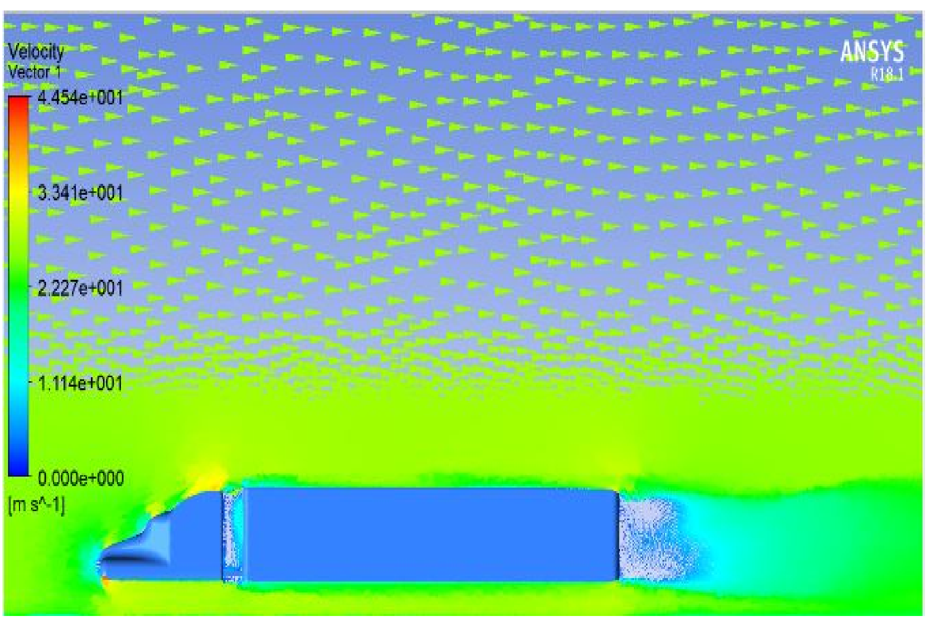


Fig.16Velocity vectors (without Airtabs)

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue IX, September 2017- Available at www.ijraset.com

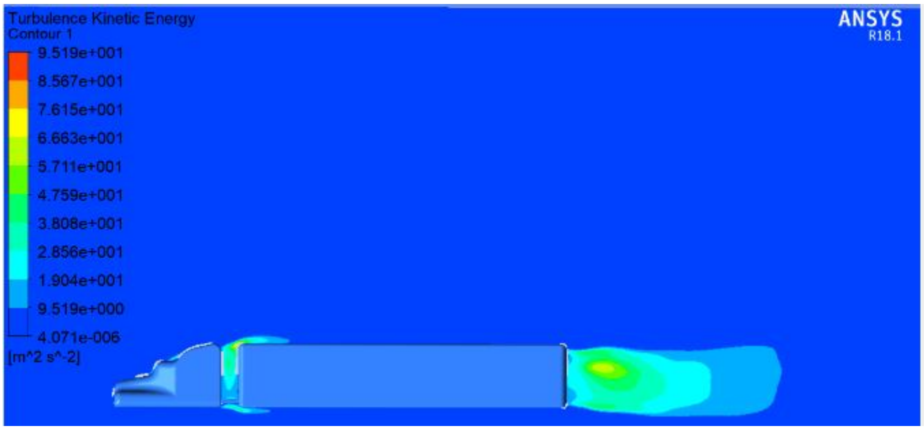


Fig.17Turbulence kinetic energy contour (without Airtabs)

B. *Vortex generator installation*

After observing the Flow around the walls of the body and closely studying the turbulence generated in wake regions, it is observed that the flow separation occurs (between tractor and trailer and at the rear end of the trailer) due to a sudden fall in the curvature of the body. Trucks, being a bluff body, account to a large boundary layer separation. Large turbulence is generated in this wake region (blue region) that develops due to flow separation. This blue color means the direction of velocity is in opposite direction or there is backflow. However, the velocity of air in wake regions is not zero but of lesser magnitude. To reduce these wake regions and hence reduce drag, the Airtabs were installed at the end of the tractor and also at the end of the trailer where the flow separates. Also, in the installation manual of the Airtabs, it is prescribed to install them as close as possible to the rear (trailing) edges of the vehicle [10].

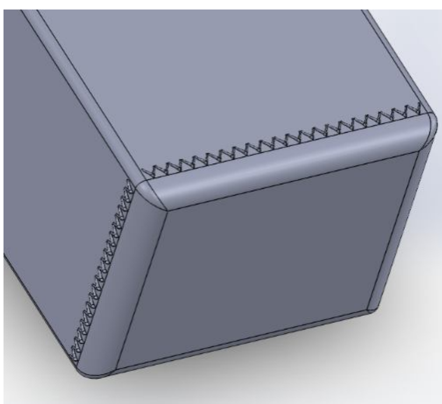
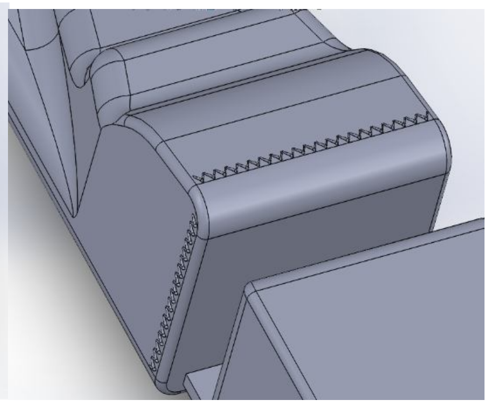



Fig.18 Airtab installation at rear

Fig.19Airtab installation at front

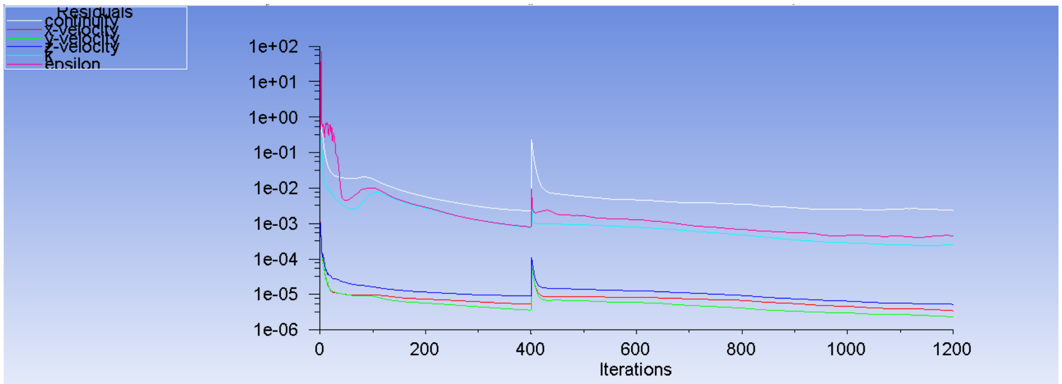
C. *After vortex generators*

Drag forces and continuity residuals were monitored as follows

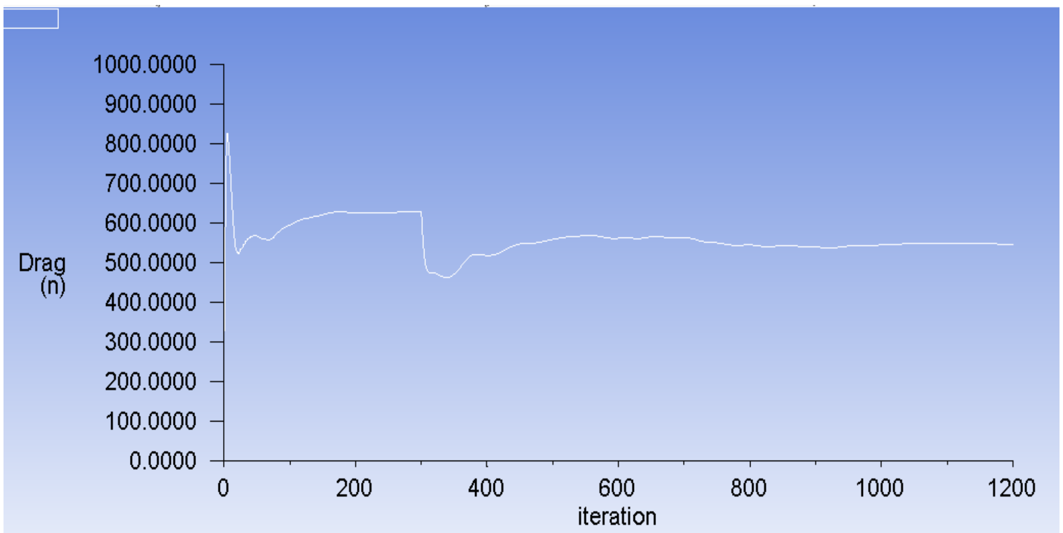
Forces - Direction Vector (0 0 -1)

	Forces (n)		
Zone	Pressure	Viscous	Total
carbody	435.94781	99.358658	535.30647
Net	435.94781	99.358658	535.30647

Fig. 20Force Report (With Airtabs)



Graph 4 Residual Curves (With Airtabs)



Graph 5 – Drag monitor (With Airtabs)

The Velocity and Turbulence Kinetic Energy contours are shown below when solution was obtained without vortex generators.

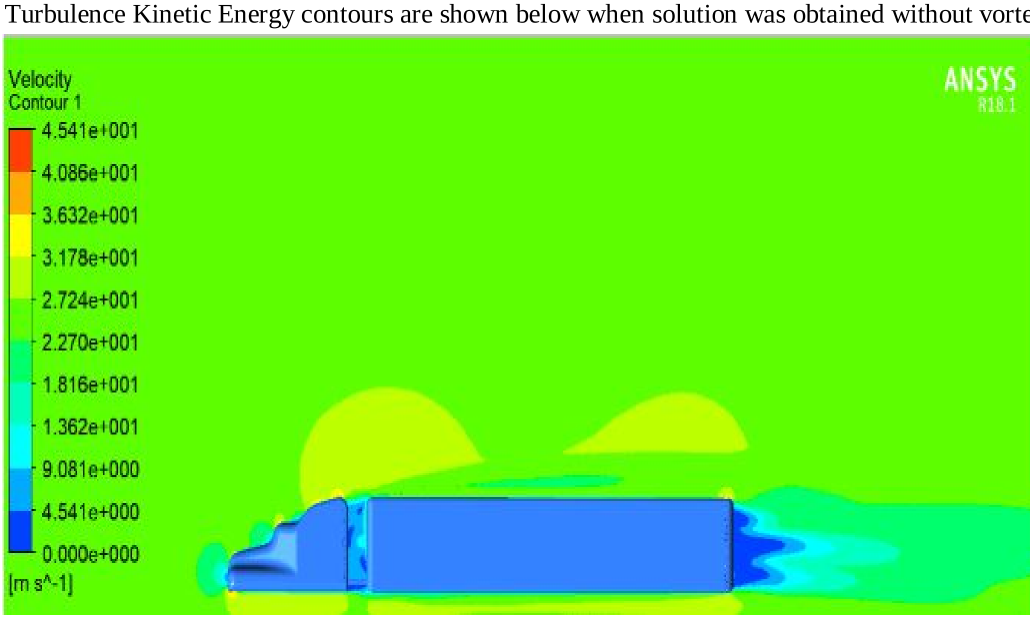


Fig. 21 Velocity Contour (with Airtabs)

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue IX, September 2017- Available at www.ijraset.com

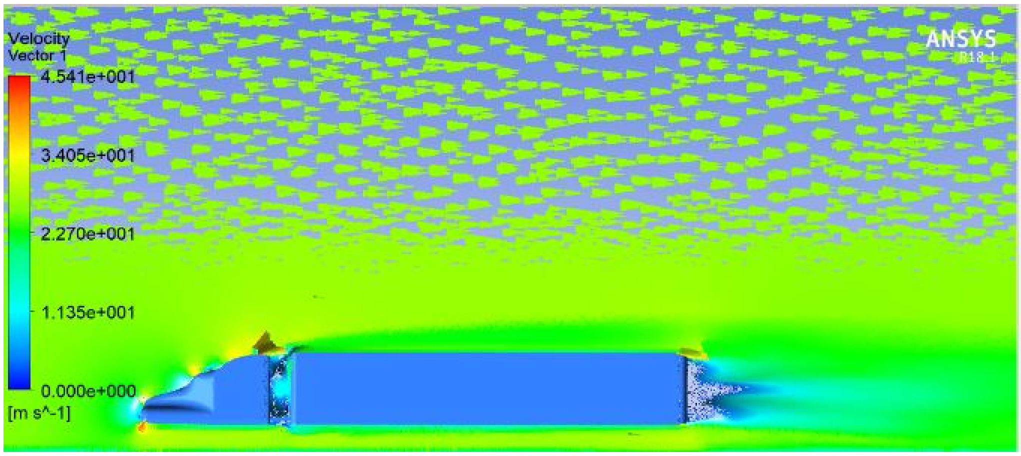
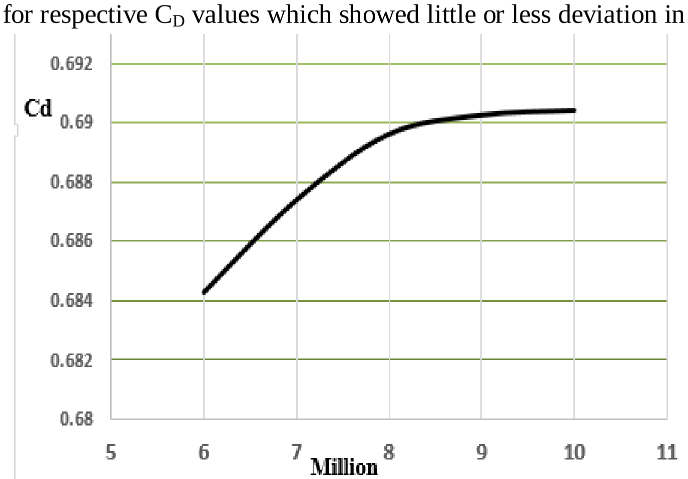


Fig.22Velocity vectors (with Airtabs)

Fig.23Turbulence kinetic energy contour (with Airtabs)

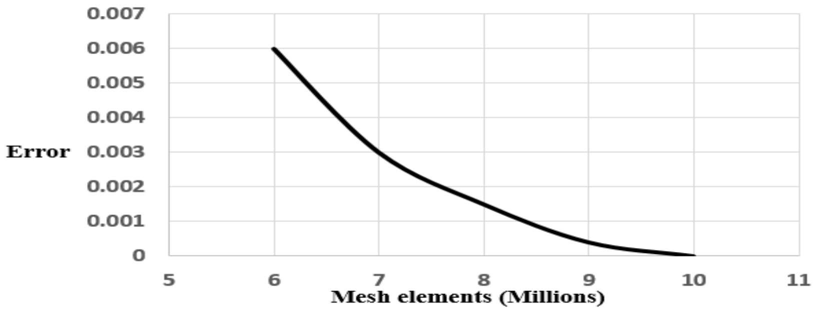
D. Mesh Independence

Experiment was performed five times with varying mesh quality (6 million, 7 million, 8 million, 9 million and 10 million mesh elements) while rest of the settings were kept unchanged. After a certain mesh refinement, there is no significant difference in the value of C_D. Graph was generated for respective C_D values which showed little or less deviation in the values of C_D.



Graph 6 – C_D Vs Mesh elements (millions)

Using the C_D value for 10 million mesh elements as reference, errors were calculated in rest of the values of C_D and an error vs. mesh elements graph was plotted.



Graph 7 – Error VsMsh elements (millions)

Slope of this line is then calculated at the point of 7 million and 8 million mesh elements and the slope came out to be -0.5 which is in accordance to the solution settings we have done by using second order equations of momentum, turbulent kinetic energy and dissipation. Therefore it proves the reliability of values we get for the drag reduction using Airtabs in our CFD simulation [14].

VII. CONCLUSION

After completion of experimentation and analysis, following drag reduction was monitored.

Table Iii
Results Of Drag Reduction

At 95 KMPH	Drag force from Fluent (N)	Actual Drag force (N)	Actual C_D
Without Airtabs	544.592	1089.184	0.6904
With Airtabs	535.306	1070.612	0.6786
% Reduction	1.705%	1.705%	1.705%

The Coefficient of drag given by Fluent is then again validated for real values with the data from the wind tunnel testing of tractor-trailer, which should be in the range of 0.5-0.9 (depending upon the aerodynamic shape of the body) [11]. As the tractor-trailer model taken for this study is already streamlined and has chamfered roof edges, a low value of drag coefficient is expected. The value of C_D determined through our simulation comes out to be 0.6786 for the whole body which is in the range specified.Thiverifies that the simulation provides result which are close to the real values.

Drag force-velocity relation can be given as: -

Force = C_D A ρ V^2

Engine Power required to overcome Drag = Force x Velocity

Power = C_D A ρ V^3

The above relation tells that reduction in C_D directly affects the power consumed by engine to overcome this drag force and this increases the fuel economy. As power is a cubic function of velocity, so only at high velocities (which is achieved at highways) the drag reduction helps prominently in fuel economy.

The percentage drag reduction with Airtabs is only 1.705% which is less than the reduction which can be achieved by using other drag reduction devices (Up to 26.1% drag reduction is possible [15]). But the fact that Airtabs are cheaper and are easier to install than other alternatives should also be taken into account.

REFERENCES

[1] Jason Morgan 'Investing in aerodynamics to improve fuel efficiency', Fleet Equipment Mag. Published 15-08-2017. Link:- http://www.fleetequipmentmag.com/truck-trailer-aerodynamics-fuel-efficiency/

[2] Christopher Michael Mugnaini (2015), 'Aerodynamic drag reduction of a tractor trailer using vortex generators: A computational fluid dynamic study'. Presented to the faculty of the Department of Mechanical Engineering California State University, Sacramento. Nov 30, 2015.

[3] J. Leuschen and K. R. Cooper. 'Full-Scale Wind Tunnel Tests of Production and Prototype, Second Generation Aerodynamic Drag-Reducing Devices for Tractor Trailers'. SAE TECHNICAL PAPER SERIES, Vols. SAE 06CV-222, 2006. DOI-10.4271/2006-01-3456

[4] M.Kanagaraj and Dr. S. Periyasamy, 'Drag reduction using vortex generator on a passenger vehicle'. International Journal of Innovative Research in Science, Engineering and Technology, Volume 6, Special Issue 7, April 2017, ISSN.

[5] Mechanical Engineering Department, 'Turbulent wake flow around circular cylinder'. Florida State University, College of Engineering, September 9, 2012. Link:https://www.eng.famu.fsu.edu/~shih/succeed/cylinder/cylinder.htm#Other%20Useful%20Links

[6] 'Airtabs, how do they work?' Aeroserve Technologies Ltd. December 05, 2014. Link: - http://www.airtab.com/how-do-they-work.htm

[7] Ansys Fluent Theory Guide 12.0, Release 12.0 © ANSYS, Inc. 2009-01-23

[8] T.S.D.Karthik, 'Turbulence models and their applications', Department of Mechanical Engineering IIT Madras. Guide: Prof. Franz Durst

[9] FLUENT 6.3 User's Guide, © Fluent Inc. 2006-09-20

[10] 'Airtab® installation instructions', Aeroserve Technologies Ltd. August 2009.

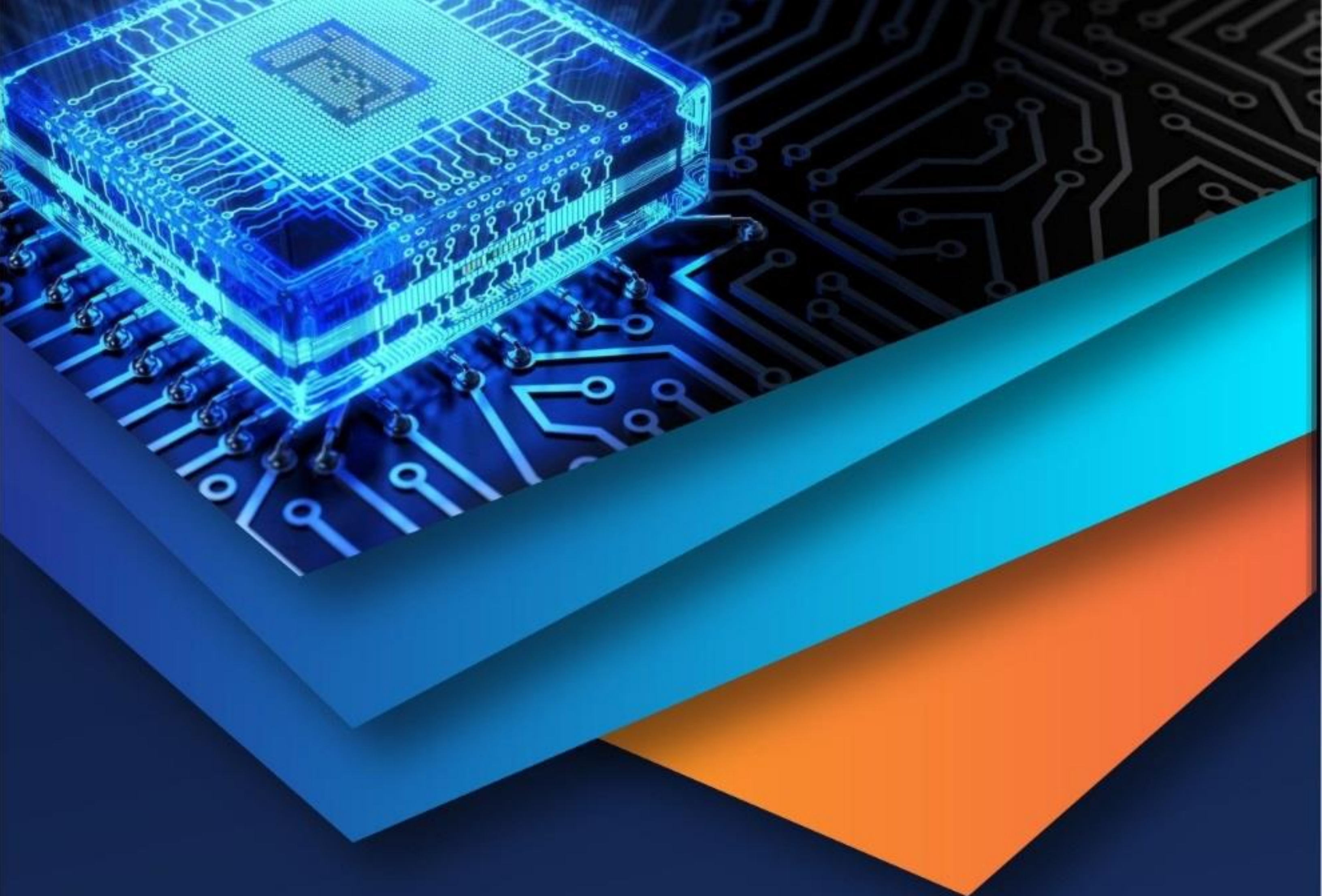
[11] Gino Sovran, 'Aerodynamic drag mechanisms of bluff bodies and road vehicles' (1978). Springer Science & Business Media, 06-Dec-2012.

[12] Introductory FLUENT Notes FLUENT v6.3 December 2006, Fluent User Services Center

[13] Vijay Mali, SanketDange, 'Basic of y plus, boundary layer and wall function in turbulent flows', 25 July. Link: https://www.learncax.com/knowledge-base/blog/by-category/cfd/basics-of-y-plus-boundary-layer-and-wall-function-in-turbulent-flows

[14] Nobuyuki Satofuka, 'Computational fluid dynamics 2000: proceedings of the first international conference of computational fluid dynamics' –Kyoto, Japan. Published May 14-17, 2018.

[15] HarunChowdhury, HazimMoria, Abdulkadir Ali, Iftekhar Khan, FirozAlam and Simon Watkins , 'A study on aerodynamic drag of a semi-trailer truck'. Procedia Engineering. School of Aerospace, Mechanical and Manufacturing Engineering, RMIT University, Melbourne, VIC 3083, Australia. Procedia Engineering 56 (2013) 201 – 205.



10.22214/IJRASET

INDEX COPERNICUS

45.98

IMPACT FACTOR: 7.129

TOGETHER WE REACH THE GOAL

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY