

# Study Of Air Flow Over the Truck Using Open Foam

<sup>#1</sup>N.A.Nawale, <sup>#2</sup>Dr. S.L.Borse

<sup>1</sup>nawalenarendra@gmail.com

<sup>2</sup>sachinlb@yahoo.co.uk

<sup>#12</sup>Department of Mechanical Engineering, Rajarshi Shahu College of Engineering,  
Pune, India.



## ABSTRACT

In this current study the hybrid mesh and the  $k-\omega$  turbulence model were adopted to investigate the variations of the aerodynamic loads and the flow field of heavy-duty truck while moving on the road with the velocity of 40 m/s, 50 m/s, 60 m/s, 70 m/s and 80 m/s. The results showed that, with the variation of the gap between cabin and trailer the forces acting on truck are changes. Drag Force and Lift Force are compared against the different combinations of the cabin and trailer using open source CFD Software OpenFOAM. Geometry was made in CATIA. Meshing was done using unstructured mesh tool of OpenFOAM called as snappyHexMesh. Steady incompressible flow simulations were performed for a modified full scale version of the geometry to evaluate drag reduction.

**Keywords-**  $k-\omega$  turbulence model, Drag Force, Lift Force, Incompressible flow, OpenFOAM

## ARTICLE INFO

### Article History

Received : 18<sup>th</sup> November 2015

Received in revised form :

19<sup>th</sup> November 2015

Accepted : 21<sup>st</sup> November ,  
2015

**Published online :**

**22<sup>nd</sup> November 2015**

## I. INTRODUCTION

In this project mainly concentration is given to numerical investigations carried out on flow of air the truck model. Comparing with different combinations of the truck at speed range from 40 km/hrs to 80 km/hrs. Here evaluation of coefficient of drag, coefficient of lift, drag force, lift force are done by OpenFOAM Software. OpenFOAM is a C++ library, used primarily to create executables, known as applications. The applications fall into two categories: solvers, that are each designed to solve a specific problem in continuum mechanics; and utilities, that are designed to perform tasks that involve data manipulation. OpenFOAM is supplied with pre- and post-processing environments. blockMesh, snappyHexMesh, paraView. The interface to the pre- and post-processing are themselves OpenFOAM utilities, thereby ensuring consistent data handling across all environments. Every OpenFOAM case must contain three folders, 0, system and constant. 0 folder contains initial condition and boundary condition. System folder contains all the solver settings and simulation control, eg. solver to be used, time steps, force calculation(if any), discretization schemes etc. Constant folder contains mesh information in native format, polyMesh. It also contains turbulence model to be used, flow properties eg. kinematic viscosity. OpenFOAM have a rich parallel implementation of almost every application. This considerably reduces the simulation time. The parallel running uses the public domain OpenMPI

implementation of the standard message passing interface (MPI).

### Problem Definition:

The current truck model is under study. There are two cases, one is truck with wheel and another case is truck without wheel. For both the cases we are going to change the gap between cabin and trailer portion and accordingly we can study the both cases for different modes of speed against different gap between the cabin and the trailer.

## II. METHODOLOGY

The OpenFOAM (Open Field Operation and Manipulation) CFD toolbox is used to analyze the air flow through the geometries for both close and open window Bus. OpenFOAM has extensive range of features to solve anything complex fluid flow involving chemical reactions, turbulence, and heat transfer to solid dynamics and electromagnetic. The Truck with various combination of gap between cabin and trailer is been modeled in CATIA V5 R16. Then model is saved in .stl extension file. The domain is defined in which Truck is to be placed. STL file is then linked to snappyHexMesh for generation of mesh. For the generation of initial flow we are using potentialFoam. By giving initial boundary conditions calculations are done by

using simpleFoam and finally post processing is done by using paraFoam.

The velocity and pressure distribution over the geometries are visualized and discussed. Various combination of the truck with respect to the gap between cabin and trailer are analyzed for different speeds of the truck. Then accordingly coefficient of drag, coefficient of lift, drag and lift force are calculated.

**III. THEORY**

*Drag Force (F<sub>D</sub>):*

Aerodynamic drag force is the force acting on the vehicle body resisting its forward motion. This force is an important force to be considered while designing the external body of the vehicle, since it covers about 65% of the total force acting on the complete body. The drag coefficient is dimensionless quantity that describes vehicle aerodynamic resistance and is useful tool for comparing different vehicle shapes. The Aerodynamic drag force is calculated by the following formula:

$$F_D = \frac{1}{2} \rho V^2 C_D A$$

where:

F<sub>D</sub> = Drag Force

C<sub>D</sub> = Drag Coefficient

A = Frontal Area Of The Vehicle

V = Wind Velocity, ρ = Air Density

*Lift Force (F<sub>L</sub>):*

Lift force causes the vehicle to get lifted in air as applied in the positive direction, whereas it can result in excessive wheel down force if it is applied in negative direction. Engineers try to keep this value to a required limit to avoid excess down force or lift.

The formula usually used to define this force is written as:

$$F_L = \frac{1}{2} \rho V^2 C_L A$$

where:

F<sub>L</sub> = Lift Force

C<sub>L</sub> = Lift Coefficient

A = Frontal Area Of The Vehicle

V = Wind Velocity

ρ = Air Density

For aerodynamic simulations, it is important to obtain accurate results for the drag and lift values. In this section the drag and lift obtained by OpenFOAM are presented. The calculations are carried out for 300 iterations.

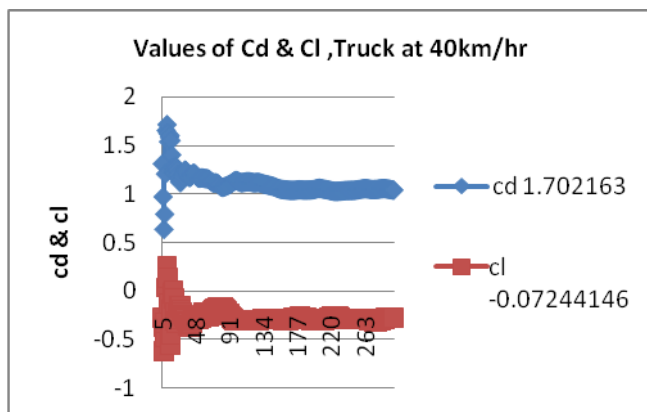


Fig 3.1 Drag & Lift Coefficient vs Number of Iterations

from the above graph It is conclude that at the initial stage of the iterations the values of C<sub>D</sub> and C<sub>L</sub> are higher but as the iterations are progresses values of C<sub>D</sub> and C<sub>L</sub> will become nearly constant.

*Case Details:*

In this section discussion is made on the various cases that are considered for next calculation and simulation work.

The cases discussed are as follow

1. Truck with wheel
  - a) Gap variation
  - b) Cover gap
2. Truck without wheel.
  - a) Gap Variation
  - b) Cover Gap

Actual dimensions of Truck

Overall length of Truck - 9.9m

Overall width of Truck– 2.3m

Overall height of Truck – 3.2m

Overall gap size between cabin and trailer-1m

**IV. COMPUTATIONAL SETUP**

During this project, different cases were considered in order to achieve the given goals. However, the simulation setup is the same for all the cases and hence the procedure is same for all cases. The first step required to begin a new simulation is to input the actual mesh into the file structure of OpenFOAM .But in our project we are creating the geometry in CATIA V5 modeling software. It is then converted into the Stereolithography (.stl) file.

*Domain Specifications:*

The computational domain is designed to lead to a free flow with neglect-able blockage, which essentially means a box that consist of a inlet, a outlet, two sides, a roof and ground surface. When the geometry was defined in the creation of the computational mesh, all faces of the domain were assigned names. The names of the inlet and outlet planes (at x = 0 and x = L) are front face and back face of domain as velocity inlet and pressure outlet respectively. The names of the planes at y = L, z=0, and x=L are outer wall as wall. The names of the model are bus as a wall.

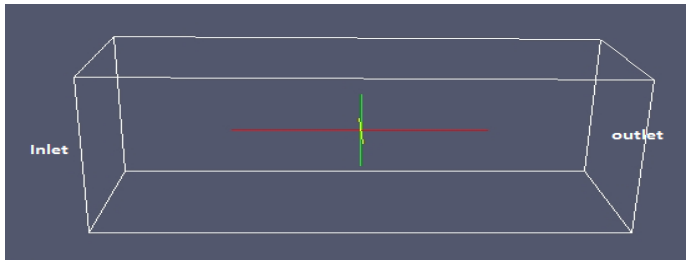


Fig.4.1 Defining the Domain

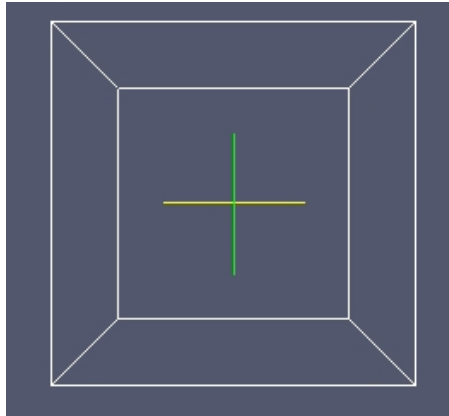


Fig.4.2 F.V. Without geometry

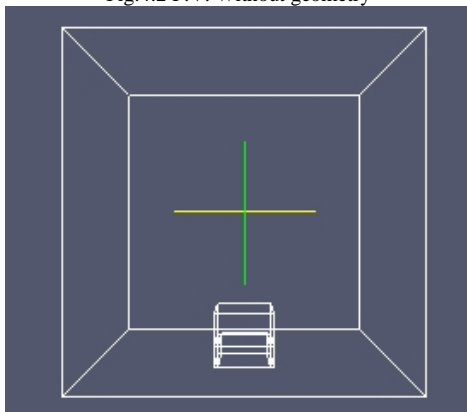


Fig.4.3 F.V. With geometry

Dimensions of the domain:

Height = 10m

Length = 24 m

Breath = 10m

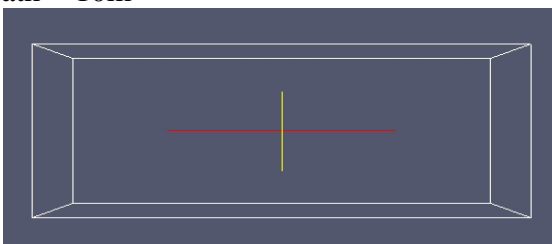


Fig.4.4 Domain Without Geometry

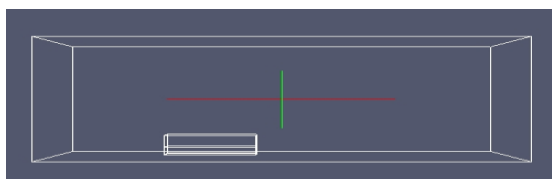


Fig.4.5 Domain With Geometry

Mesh:

Mesh is created using snappyhexmesh, OpenFOAM's native meshing tool. Utility snappyHexMesh is used to create high quality hex dominant meshes based on arbitrary geometry. It is controlled by parameters in the le snappyHexMeshDict. It can be executed in parallel. It preserves the feature edges, addition of wall layers. Details of mesh size is given in the following table 5.1

TABLE I : DETAILS OF MESH SIZE

No.of points	No.of cells	No.of faces	No.of internal faces
2185655	1986273	6157320	5963048

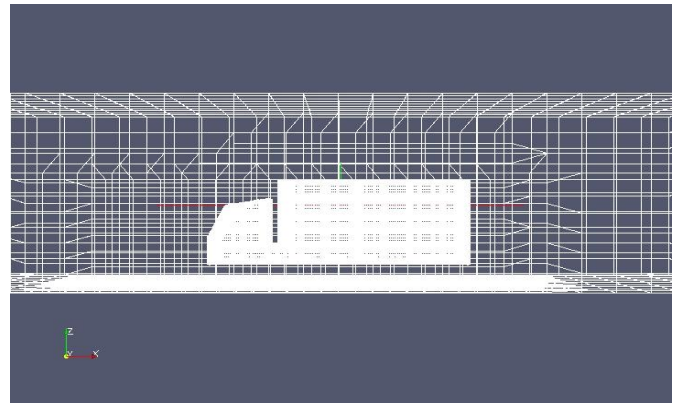


Fig.4.6. Front View

Boundary conditions: In project the tests are carried out at different operating speed range such as 40km/hr,50km/hr,60km/hr,70km/hr,80km/hr. That means each geometry of Truck can pass through these speed range. For example consider case 1 when it pass through 40km/hr, velocity of the inlet boundary condition is set with value 40km/hr (11.11m/s) with temperature of 300K (26.85°C). The outlet boundary condition is set to pressure outlet with gauge pressure of 0 pa. The density of the air is set as 1.125kg/m<sup>3</sup> and the viscosity of the air is 1.7894×10<sup>-5</sup> kg/(ms).

Turbulence Modeling:

All of the simulations took turbulence into account with the k- $\omega$ -SST turbulence model. This model was used for its  $\omega$  proven reliability in separation zones and its ability to blend a good free stream model to a good boundary layer model.

Turbulence Intensity: The turbulence intensity,  $I$ , is defined as the ratio of the root-mean-square of the velocity fluctuations,  $u'$ , to the mean free stream velocity,  $u$ .

$$I = \frac{u'}{u}$$

For internal flows the value of turbulence intensity can be fairly high with values ranging from 1% - 10% being appropriate at the inlet. The turbulence intensity at the core of a fully developed duct flow can be estimated as:

$$I = 0.16Re^{-1/8}$$

For external flows the value of turbulent intensity at the free stream can be as low as 0.05% depending on the flow characteristics. We are considering turbulence intensity as 0.02%.

About Turbulence Length Scale: The turbulence length scale,  $l$ , is a physical quantity which represents the size of the large eddies in turbulent flows. Empirical relationship

between the physical size of the obstruction (or characteristic length),  $L$ , and the size of the eddy,  $l$ , can be used to get an approximate length scale.  
 $l = 0.07L$

**Turbulent Kinetic Energy(K) and Specific Dissipation Rate ( $\omega$ )** : Required Turbulent Kinetic Energy(K) and Specific Dissipation Rate ( $\omega$ ) can be find out from the following equations and table no.II gives the (K) and ( $\omega$ ) for our selected operating speed range.

$$\text{Turbulent Kinetic Energy(K)} = \frac{3}{2} (UI)^2$$

$$\text{Specific Dissipation Rate } (\omega) = \frac{K^{1/2}}{C\mu^{1/4} \times l}$$

TABLE.II: VALUES OF K AND  $\omega$

Speed km/hr	Speed m/sec	K	$\omega$
40	11.11	0.074	0.7098
50	13.88	0.1155	0.8862
60	16.66	0.1665	1.0644
70	19.44	0.2267	1.2418
80	22.22	0.2962	1.4195

**V. RESULT AND DISCUSSION**

In this section we are going to discuss following things:  
 1)Case 1 The Truck with wheel is taken in consideration and tested at different operating speed range such as 40km/hr, 50km/hr, 60km/hr, 70km/hr, 80km/hr. The gap between Cabin and Trailer changed in the interval of 200mm from 1000mm to 0mm. There pressure contour, velocity contour, vector plot , front view, side view, and top view are pressure variation is observed.  
 2)Case 2 The Truck without wheel is taken in consideration and tested at different operating speed range such as 40km/hr, 50km/hr, 60km/hr, 70km/hr, 80km/hr. The gap between Cabin and Trailer changed in the interval of 200mm from 1000mm to 0mm. There pressure contour, velocity contour, vector plot , front view, side view, and top view are pressure variation is observed.

Case 1At speed 40Km/hrs:

Observations of various contours:

**Pressure Contour:** It is obvious from all the pressure contours that there is a higher pressure concentration on the front of the truck. Particularly, the air slows down when it approaches the front of the truck and results in that more air molecules are accumulated into a smaller space. Once the air stagnates in front of the truck, it seeks a lower pressure area, such as the sides and top of the truck.

**Velocity contour:**It is conclude that from all the velocity contours that air velocity is decreasing as it is approaching the front of the truck, then air velocity increases away from the truck front.

**Vector plot for K.E:**The red vectors has maximum kinetic energy while blue vectors has minimum. As the speed

increase the magnitude of K.E also increases and intensity of Turbulence is also increases over the truck.

**Truck Body Pressure Variation:** From the front ,side and top view we can easily observe pressure variation, the pressure is maximum at the front and minimum at the top of Truck.

TABLE III: COEFFICIENT OF DRAG AND COEFFICIENT OF LIFT FOR TRUCK WITH WHEEL, GAP BETWEEN CABIN AND TRAILER IS 1000MM

Speed Km/hrs	Cd	Cl
40	1.0452	0.2713
50	1.0440	0.2751
60	1.0454	0.2785
70	1.0434	0.2778
80	1.0462	0.2791

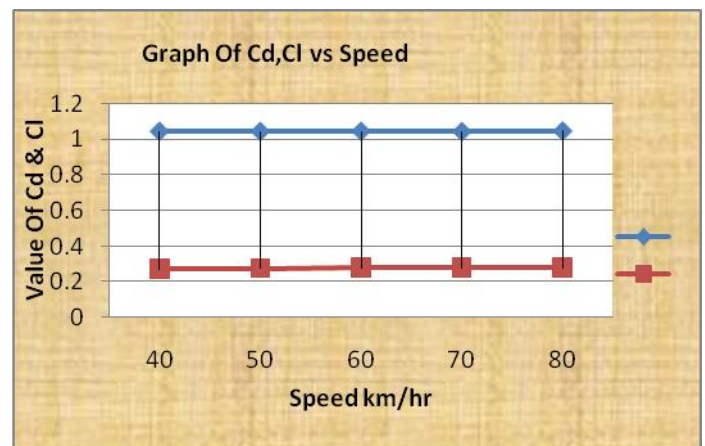


Fig5.1  $C_D, C_L$  vs Speed

Observation:

Value of Coefficient of Drag remain almost same irrespective of Speed.

Value of Coefficient of Lift fluctuate and decreases with respect to Speed.

TABLE IV: COEFFICIENT OF DRAG AND DRAG FORCE FOR TRUCK WITH WHEEL, GAP BETWEEN CABIN AND TRAILER IS 1000MM

Speed Km/hrs	$C_D$	Drag Force N
40	1.0452	626.2691
50	1.0440	976.4076
60	1.0454	1408.586
70	1.0434	1914.208
80	1.0462	2507.571

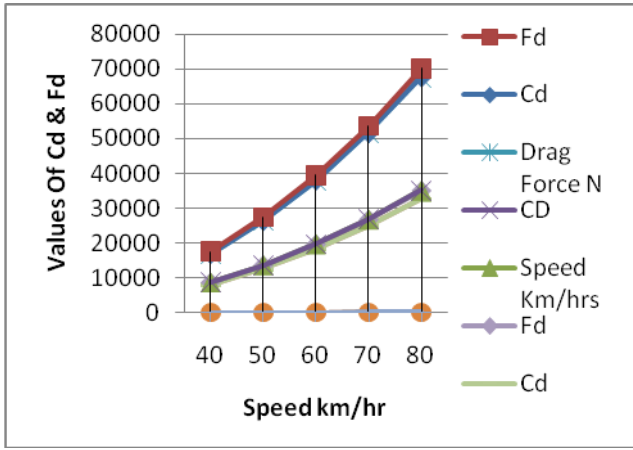


Fig4.2  $C_d$ , Drag Force V/S Speed

TABLE V: COEFFICIENT OF LIFT AND LIFT FORCE FOR TRUCK WITH WHEEL, GAP BETWEEN CABIN AND TRAILER IS 1000MM

Speed Km/hrs	$C_l$	Lift Force N
40	0.27134	162.5837
50	0.2751	257.2736
60	0.27856	375.3174
70	0.27785	509.7177
80	0.27911	668.9459

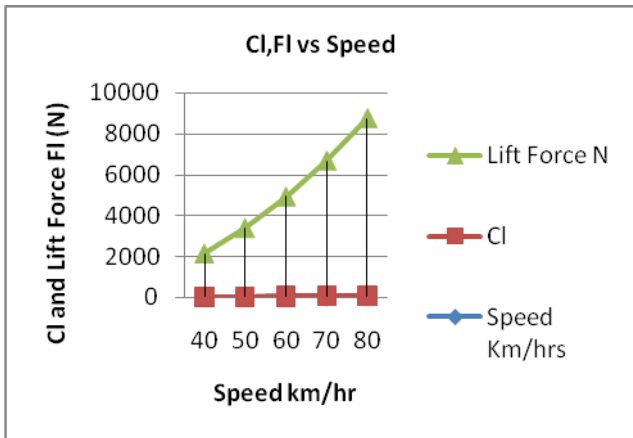


Fig 5.3  $C_L$ , Lift Force v/s Speed

*Observation* : Drag Force increases with speed not due to coefficient of Drag but due to square of Speed. Lift Force increases with speed due to combination of fluctuation of value of Coefficient of Lift and Speed.

TABLE VI : VALUE OF K,P AND U TRUCK WITH WHEEL, GAP BETWEEN CABIN AND TRAILER IS 1000MM

Speed(km/hr)	K	P	U
40	8.6255	37.6536	16.75
50	13.3732	58.6971	20.9218
60	19.2965	84.5092	25.1483
70	26.3849	115.2131	29.4214
80	34.5081	150.3068	33.4907

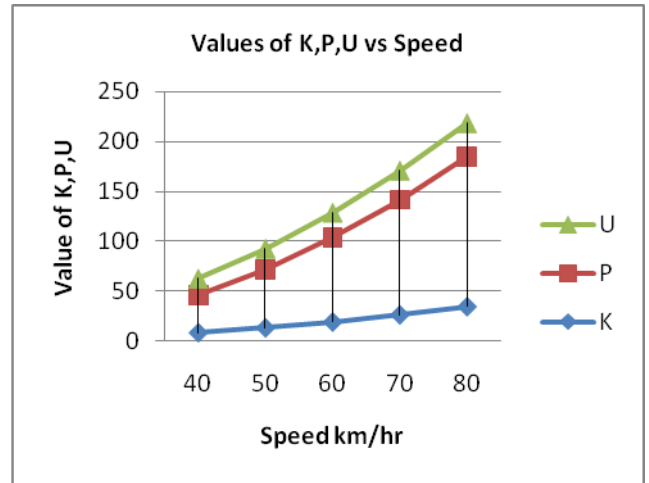


Fig5.4 K,P,U vs Speed

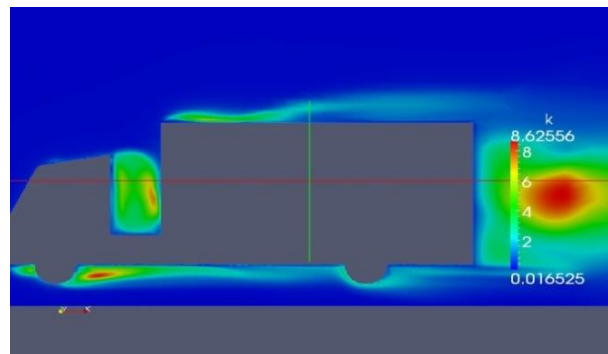


Fig.5.5 Vector Plot For K E

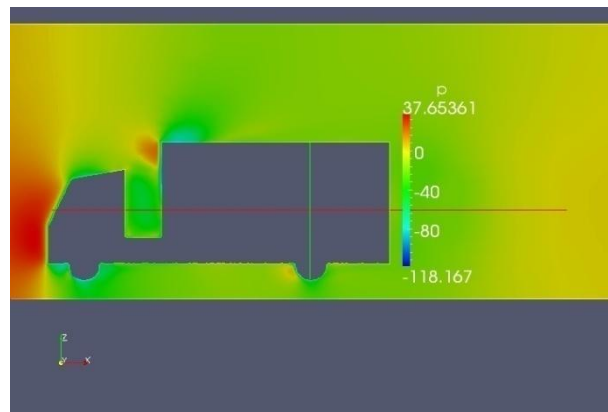


Fig5.6 Pressure Counter

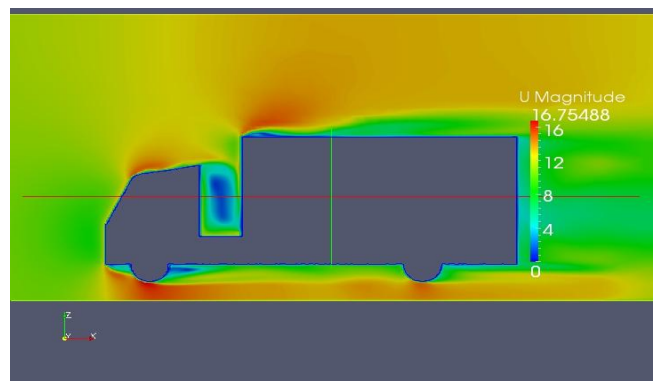


Fig.5.7 Velocity Contour



*Pressure contour:* It is obvious from all the pressure contours that there is a higher pressure concentration on the front of the Truck. Particularly, the air slows down when it approaches the front of the Truck and results in that more air molecules are accumulated into a smaller space. At the front top portion of the trailer pressure concentration is there.

*Velocity contour:* It is concluded that from all the velocity contours that air velocity is decreasing as it is approaching the front of the truck.

*Vector plot for K.E:* The red vectors has maximum kinetic energy while blue vectors has minimum. As the speed increase the magnitude of K.E also increases and intensity of Turbulence is also increases at top of truck.

*Truck Body pressure variation :* From the front ,side and top view we can easily observe pressure variation, the pressure is maximum at the front and minimum in side and top of truck

## VI. CONCLUSION

From paper it is concluded that as speed increases drag force also increases .We have studied the truck model in case I and results analysis done for different speed. To overcome increased drag we can maintain the distance between cabin and trailer. Ultimately it can reduce the fuel consumption and emissions.

## II. ACKNOWLEDGMENT

The authors would like to present their sincere gratitude towards the Faculty of Mechanical Engineering in Rajarshi Shahu College of Engineering, Pune.

## REFERENCES

- [1] Schoon, R. E., 2007. On-road Evaluation of Devices to Reduce Heavy Truck Aerodynamic Drag. SAE Publication, SAE Paper No. 2007-01-4294.
- [2] Cooper, K. R., 2006. Full-Scale Wind Tunnel Tests of Production and Prototype, Second- Generation Aerodynamic Drag-Reducing Devices for Tractor-Trailers. SAE Publication, SAE Paper No. 2006-01-3456.
- [3] Zulffaa,K. , Antonio,F, " Fuel Saving on a heavy Vehicle via Aerodynamic Drag Reduction", Elsevier Journal ,UK,PP(275-287),2010
- [4]Fred Browand , Scott Johnston, " DOE Project on Heavy Vehicle Aerodynamic Drag ", Heavy Vehicle Systems Optimization Program,PP(4-8),2004.
- [5] McCallen, R., Couch, Bell, J., Steve, W., Zilliac, G., 2000. " Aerodynamic drag of heavy (class 7-8): Simulation and benchmarking," SAE Paper 2000-01-2209 (2000).
- [6] Hucho, W, H., 1998. "Aerodynamics of road vehicles," Fourth Edition, SAE International, 1998.

[7] JUSTIN MORDEN, "Aerodynamic Analysis And Testing Of A Formula Renault Race Vehicle Using Analytical And Computational Methods Validated By Real World Test Data.

[8]Shailesh Jindal , "Numerical Investigation of Road Vehicle Aerodynamics Using Immersed Boundary RANS Approach",2005 SAE International

[9]Cooper, K. R. 2004. Commercial Vehicle Aerodynamic Drag Reduction: Historical Perspectives as a Guide. In The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains (McCallen, R., Browand, F., and Ross, J., eds.) pp. 9- 28, Springer, New York.