NUMERICAL INVESTIGATION OF DRAG ON A TRAILING AERODYNAMIC SEDAN VEHICLE

ADARSH P[1], ANDREW GEORGE CHERIAN[2], ARUN K S[3], D ROSHAN KUMAR[4]

[1,2,3,4]T.K.M College of Engineering, Kollam, Kerala, India
E-mail: adarshp2492@gmail.com, cyrilcyrilandrew@gmail.com, arunkstcr@yahoo.com, roshankumar27@yahoo.com

Abstract- The study focusses on the slipstreaming effect on an aerodynamic sedan vehicle, trailing a container truck on highways. The simulation was carried out using ANSYS FLUENT CFD software. The effect of distance of separation of the vehicles to slipstreaming has been studied and critical distance is found out. The relative drag during slipstreaming at different velocities at the critical distance has also been addressed. f reckless driving in sensitive areas.

Index terms- Pressure drag, slipstreaming, trailing vortices, wake region

I. INTRODUCTION

Drag reduction has always been the concern of any aerodynamic design. As far as ground vehicles are concerned, 90% of the overall drag is contributed by the pressure differences in front and rear end of the vehicle which is called pressure drag [1]. This is due to the separation of flow at the trailing edge of the vehicle are due to turbulence and vortices. This produces a low pressure turbulent region behind the vehicle, called wake. As the vehicle moves forward the wake also moves with it. The movement of air behind the vehicle is termed as slipstream. Many methods has been developed to reduce this pressure drag. Spoilers can be used to reduce the pressure drag but it adds to the skin friction drag hence reduction in overall drag is not appreciable. The utilization of the slipstream of a leading vehicle and hence reducing the drag of trailing vehicle is termed as slipstreaming. Slipstreaming was first observed among birds. Group of birds flying in a specific pattern reduces their overall drag force. The bird leading will experience more drag and hence needs more effort than trailing birds. Cyclists also utilize each other's slipstream in order to reduce the overall drag on the system and hence reduce the physical exertion. Utilization of slipstreaming effect can be crucial factor in any speed races. This method of drag reduction is effective only when high speed travelling is considered on highways which will improve the trailing vehicle's performance. The flow around wake of an automobile is an interesting phenomenon and many experiments and analysis were conducted in favor of this. Ahmed (1981) performed a series of wind-tunnel experiments in order to examine the wake structure around typical automobile geometries. The study focused on the time averaged structure obtained from visualizations of flow in the wake region for smooth quarter scale automobile models. Experiments were also performed with a bluff- body, "generic" vehicle geometry where pressure measurements, wake surveys and force measurements for different angles of base-slant are presented. Results indicated that almost 85% of total aerodynamic resistance is contributed by pressure drag and most of this drag is generated at the rear end.

Han (1989) performed a numerical study over a three-dimensional bluff-body in proximity to the ground. Due to the lack of detailed velocity measurements over this body, this work compared qualitatively the formation of the vortices in the wake and the drag coefficient for different slant angles. The author used two different RANS based turbulence models namely the k-ε and the RNG k-ε. It was reported that the overall validity of the computations was dependent on the turbulence model and the accuracy of the discretization scheme.

Sinisa and Davidson (2003) stated that the consideration was not only the drag and lift coefficients that describe the aerodynamic properties of the body, but also the flow structures responsible for these properties. The forces acting on the surface of the body as a result of the surface pressure were studied both time averaged and instantaneously in the study. So, while the flow around the bodies is highly unsteady, knowledge of this flow is based primarily on experimental and numerical studies of time averaged observations. The study has shown that the instantaneous flow is very different from the time averaged one, not only in the wake region but also along the entire body.

Gokul Krishnan Rajamani (2006) studied time-averaged characteristics of a simplified, generic passenger vehicle, called the Ahmed car model. The 2D analysis of isolated model showed that the wake structure influences the drag coefficient of the model which is dominated by rear slant angles of the Ahmed body. Significant drag reduction was observed for 25˚ rear slant angle, due to reduced recirculation region behind the model.

Shahrin Hisham Amirnordin et al. (2010) performed a study which focusses on the drag and lift forces on a hatchback car trailing a large vehicle. He studied the effect of distance in the drag of a body at a Reynolds number of 3.65x105. He found out that aerodynamically optimum critical distance was between 4 to 5 m. Martin Bjerling et al (2011) conducted large eddy simulation (LES) on a vehicle platoon of four boxes.

Simulations, performed on a static four box platoon and static single box, were compared with wind tunnel tests. Simulations where one box in the platoon was oscillating were also compared with corresponding wind tunnel experiments. The change in drag for the platoon and the resulting hysteresis effect was studied. The calculated CD values for boxes 2, 3 and 4 are in good agreement with the experimental results.

The review of the literature shows that the slipstreaming effect on aerodynamic sedan has not been addressed.

The present study focusses on the slipstreaming analysis on aerodynamic sedan, trailing a large container truck on highways, i.e. at higher speeds. The effect of leading vehicle on the overall drag of the trailing vehicle at different speed of travel has also been addressed.

II. METHODOLOGY

2.1. ANALYSIS METHOD

The simulation has been done using ANSYS-14. FLUENT solver was used to solve the Navier-Stokes equation. In order to study the effects of vehicle platooning one can either analyze the flow in a wind tunnel or Computational Fluid Dynamics (CFD). A CFD simulation is preferable to do instead of a wind tunnel experiment due to the much lower cost. It is also preferable to perform CFD analysis as aerodynamic problems become more complex. A steady measurement of the drag force on a car might still be easily achieved in a wind tunnel, however analyzing flow patterns in a wind tunnel can be difficult. Thus CFD simulations are usually used for in-depth analysis of a flow, while wind tunnel experiments are more often used to confirm simulated results. In the present vehicle, CFD analysis has been performed to study the effect of slipstreaming.

2.2. MODELLING

The present study focus on 2-D analysis of sedan, trailing a container truck at different speeds. Initially a CAD model of a sedan car was made as per the dimensions specified by the manufacturer. The model was 1:1 scale of the original automobile. The CAD model is shown in Fig 1. From this model the car has been imported to Solidworks software and the symmetrical section of the car is remodeled. The vehicle has an overall length of 4.5 m. The model was then saved as an IGES file. It is then imported to ANSYS and Boolean subtraction is done to obtain model as shown in Fig 2.

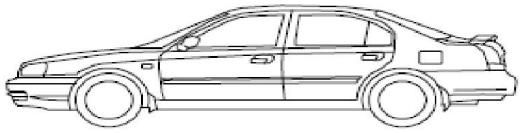


Fig 1: CAD model of the aerodynamic sedan



Fig 2: Image of modelled car in the domain

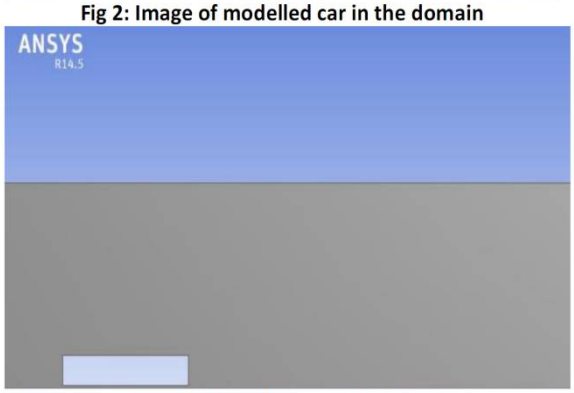


Fig 3: Image of modelled truck in the domain

As seen in the Fig 2, the symmetric section of the car is modelled (i.e. without wheels). This is done to clearly visualize the pressure variation effect in the wake region. The container truck was modelled in the ANSYS DESIGN MODELER. As far as the dimension of truck is considered only height and length are important. The height of truck is 2.5 m and length is 14 m. The image of modelled container truck is shown in Fig 3.

The domain length is taken as 250 m x 1000 m. The domain length was found out through trial and error method. After trying many combination and verifying the flow pattern domain length was fixed. For analysis of slipstream the truck was initially placed at a distance of 10 m downstream of inlet. The distance between car and truck is varied from n to 7n, where n is 4m.

2.3. MESHING

Initially car and truck was separately meshed and the CFD analysis was done on them to validate and check the domain length. In order to get required mesh, meshing was done repeatedly, varying sizing and smoothing, until the required accuracy was obtained.

The details of the mesh of car is given in Table I. The number of elements obtained was 34985 and number of nodes was 17940. Edge meshing was done initially and then face meshing was done. Automatic meshing method was used during this. Triangular type element was used to mesh the domain.

The details of mesh of truck is given in Table II. The number of elements obtained was 31043 and number of nodes was 16541. Triangular type element was used to mesh the domain. The meshing was done similar to method discussed earlier. Automatic meshing method is used here also.

TABLE I. DETAILS OF MESH FOR CAR

RELEVANCE CENTER	Coarse
SMOOTHING	Medium
SPAN ANGLE CENTER	Fine
MIN SIZE	3.6287e-004 m
MAX FACE SIZE	0.03 m
MAX SIZE	1.0 m
GROWTH RATE	2.0
MNIMUM EDGE LENGTH	1.7614e-002 m

TABLE II.DETAILS OF MESH FOR TRUCK

RELEVANCE CENTER	Coarse
SMOOTHING	Medium
SPAN ANGLE CENTER	Fine
MIN SIZE	3.75e-003 m
MAX FACE SIZE	0.375 m
MAX SIZE	0.5 m
GROWTH RATE	1.2
MINIMUM EDGE LENGTH	2.90 m

Later on both the truck and car was placed in a single domain and meshed together. In this meshing inflation of a face close to the surface of car and truck was done in order to correctly visualize the variation in parameters of flow close to the surface bodies. The growth rate during inflation was 1.2 and maximum layers were given as 2. The number of elements were 52688 and the number of nodes were 27465. Here also triangular element is used to mesh the domain. The details of the combined mesh is given in Table III. The figure of the meshed domain of the slipstream model is a shown in Fig 4.

TABLE III. DETAILS OF MESH FOR SLIPSTREAM MODEL

RELEVANCE CENTER	Coarse
SMOOTHING	Medium
SPAN ANGLE CENTER	Fine
MIN SIZE	0.2 m
MAX FACE SIZE	0.4 m
MAX SIZE	8.0 m
GROWTH RATE	1.2
MINIMUM EDGE LENGTH	1.7614e-002

2.4. BOUNDARY CONDITION

For all analysis the RANS k-ϵ turbulence model is used. The air is assumed to be incompressible and steady as the Mach number under consideration is less than 0.3. Standard wall functions are assumed. The Navier-Stokes equation for incompressible steady flow is given below

I. CONTINUITY EQUATION

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \qquad (1)$$

II. MOMENTUM EQUATION

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\Gamma + \frac{\partial}{\partial x_j}(-\rho u_i u_j) \qquad (2)$$

where Γ is defined as

$$\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_i}{\partial x_i}\right)\right]$$

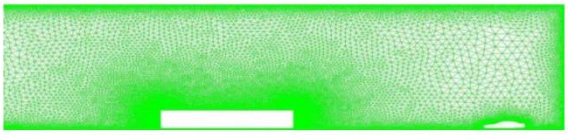


Fig 4: Image of meshed slipstream model

ρ is mean density, p is mean pressure, μ is the molecular viscosity and $(\rho u_i u_j)$ is the Reynolds stress.

2.4.1 VELOCITY INLET BOUNDARY CONDITION

Velocity inlet boundary conditions are used to define the flow velocity, along with all relevant scalar properties of the flow, at flow inlets. The stagnant properties of the flow are not constant, so they will rise to whatever value is necessary to get the required velocity distribution. The values of velocity inlet boundary conditions in FLUENT are given in Table IV.

TABLE IV.INLET BOUNDARY CONDITION

PROPERTIES	**VALUES**
Velocity Specification	Magnitude, Normal to boundary
Velocity Magnitude	80kmph,100kmph and 120kmph
Turbulence Specification Method	k-ϵ method
Turbulent Kinetic Energy	1
Turbulent Dissipation rate	1

2.4.2 PRESSURE OUTLET BOUNDARY CONDITION

Pressure outlet boundary conditions require the specification of a static (gauge) pressure at the outlet boundary. The value of static pressure specified is used only while the flow is subsonic. If the flow become locally supersonic, the specified pressure is no longer used. The default values of pressure outlet boundary conditions in FLUENT are given in Table

V. Also no slip boundary conditions are specified on two walls of the domain under consideration and the body is considered to be stationary.

TABLE V.OUTLET BOUNDARY CONDITION

PROPERTIES	VALUES
Gauge Pressure	0
Back Flow Direction Specification	Normal to Boundary
Turbulence Specification Method	k-ϵ method
Back Flow Turbulent Kinetic Energy	1
Back Flow Turbulent Dissipation Rate	1

III. RESULTS AND DISCUSSION

3.1. FLOW ANALYSIS –CAR ALONE

Initially the modelled car is located at a distance 10 m downstream from the inlet of domain. The flow is simulated for two speeds – 80kmph and 120kmph.

The pressure, velocity and vorticity vector contour at flow speed of 22.22m/s (80kmph) is plotted and the vorticity vector is shown in Fig 5. The maximum pressure obtained in this case is about 101.7kPa. This pressure is developed at a section in front of the car. From the contour it is clear that a low pressure regions formed at the rear side of vehicle. The maximum velocity attained is 38.22 m/s and maximum magnitude of vorticity vector is 1607.286 per second.

Similar values at velocity of 33.33m/s (120kmph) is also found out. The maximum value of pressure developed is about 102.1kPa. For this flow, the maximum velocity attained is 57.808 m/s. The vorticity vector obtained has a maximum value of 2440.7 per second. This is shown in Fig 6.

From the plots it has been found out that as velocity of car increases, the maximum pressure developed increases. The wake region of the car grows stronger with velocity and tail vortices also increase in magnitude. All these factors suggest that as velocity increases the drag also increases.

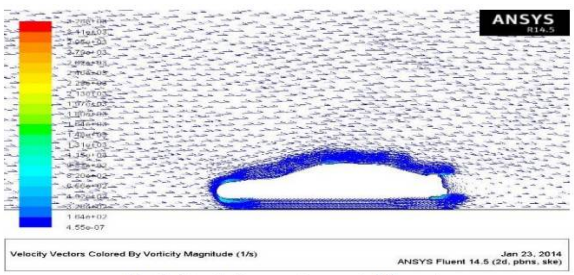


Fig 5 :Vorticity contour at 80kmph

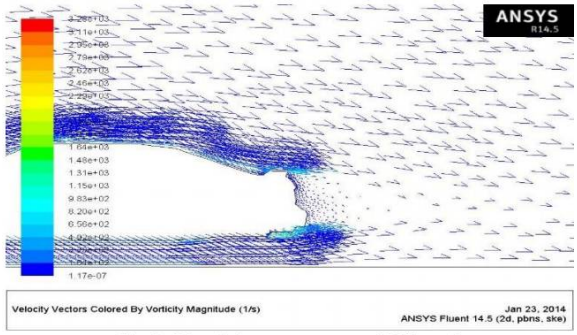


Fig 6 : Vorticity contour at 120kmph

3.2. FLOW ANALYSIS – TRUCK ALONE

The modelled truck is placed at a distance of 10 m downstream of inlet in the flow domain. The simulation is carried out at two speeds- 22.22m/s (80kmph) and 33.33m/s (120kmph). As far as the truck is considered, it is the leading vehicle in the slipstreaming analysis so the only parameter to be considered is the wake region length. So the plot of pressure with respect to distance is taken and shown in Fig 7.

In Fig 7, the pressure values at body of the truck and at the wall of domain has been plotted. From this graph we can find the wake length of the truck. The graph is magnified in order to correctly find out the position to which pressure variation is appreciable. This position gives the length of wake produced by the truck.

The Fig 8 shows similar graph at a velocity of 33.33m/s (120kmph). The wake region length was obtained in this velocity by the method discussed earlier.

It has been found out that at a speed of 22.22m/s(80kmph) the wake length produced by the truck is a bout 30m and at 33.33m/s (120kmph) it is 44 m. Thus the wake length produced is appreciable in order to utilize slipstreaming effect

The velocity and vorticity contours was also plotted in order to validate the software and domain region. The plotswere slightly different from those of car owing to the shape of truck. The vorticity vector plots at these speeds are shown in Fig 9 and Fig 10.

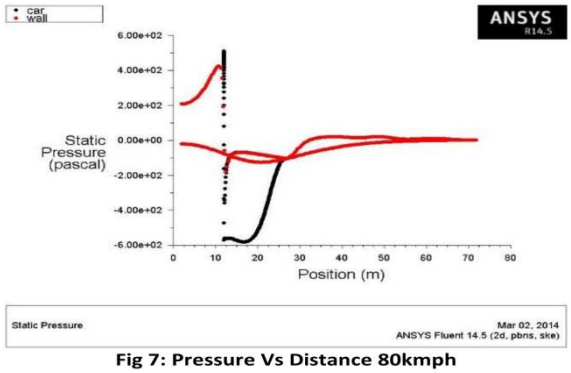


Fig 7: Pressure Vs Distance 80kmph

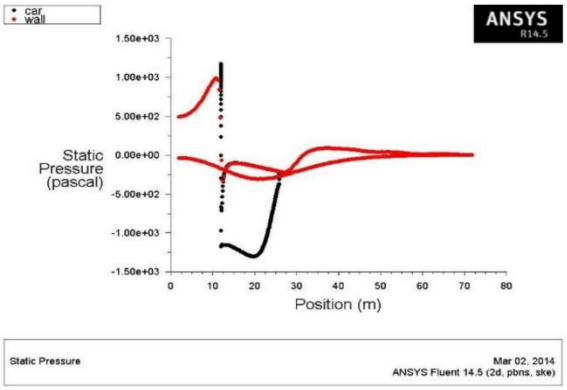


Fig 8: Pressure Vs Distance 120kmph

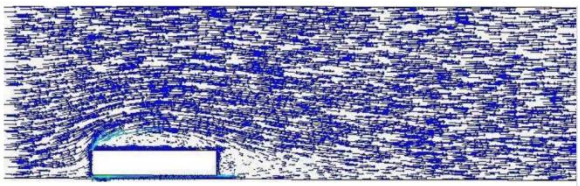


Fig 9: Vorticity profile 80kmph

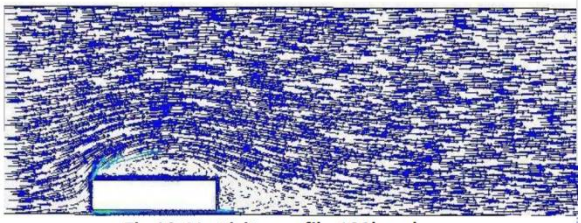


Fig 10: Vorticity profile 120kmph

At 22.22m/s (80kmph) the maximum velocity attained for the flow is 40.16 m/s and the maximum magintude of vorticity vector is 741.992 per second.

At 33.33m/s (120kmph) the maximum velocity attained is about 60.754m/s and the maximum magnitude of vorticity vector is 1105.856 per second

The vortices formed in the trailing edge of the truck will play a significant role in drag of the trailing vehicle. It is the direction of rotation of the vortex that determines whether the slipstreaming is advantageous or not. [Shahrin Hisham Amirnordin, 2010]

3. FLOW ANALYSIS- SLIPSTREAMING

The truck is located at a distance of 10 m downstream the inlet and sedan car is trailing it in the flow domain. The distance between the car and truck is then varied from n to 7n where n is 4m. The simulation is done at three speeds – 80kmph, 100kmph and 120kmph.

The forces at front and rear of car is found out assuming the area to be 1 m2. The net drag is the resultant of forces at front and rear side of the car. In order to find the net drag acting on the car the following procedure has been carried out.

I. The front end and rear end of vehicle is created as a separate entity during modelling.

II. After the analysis has been carried out, the force report option in FLUENT is used to get the forces. The force at front and rear end geometry has been found out by specifying appropriate direction.

The value of net drag force with distance between vehicles has been obtained at all the three velocities and the required graph has been plotted. From the graph the minimum drag position or optimum slipstreaming distance has been found out. The graph obtained is shown in Fig 11.

In Fig 11 as distance increases from 4m to 12m, the drag force increases as expected. But after 12m the drag on the car began to decrease. This is due to the fact that after 12m the vortex at the front of the vehicle changes direction as shown in Fig 12 and Fig 13.

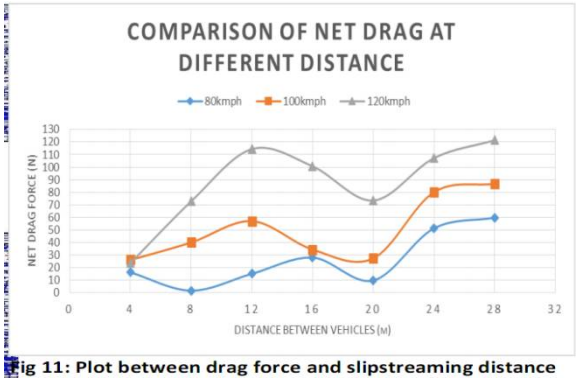


Fig 11: Plot between drag force and slipstreaming distance

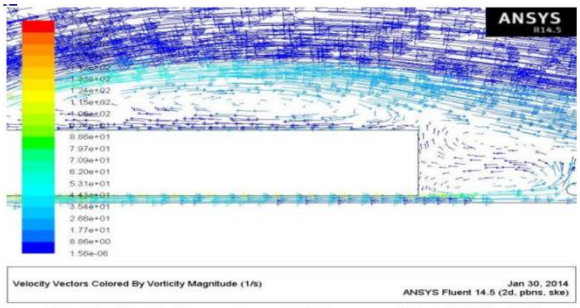


Fig 12: Vorticity vector at distance of 4m (<12m) at 80kmph

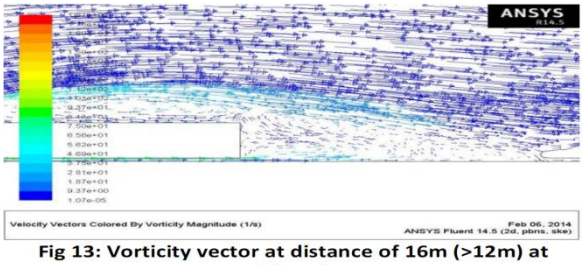


Fig 13: Vorticity vector at distance of 16m (>12m) at 80kmph

At distances less than 12m the strong trailing vortices from the downstream rotate counter clockwise through in front of the car body. The lower vortex upstream of the trailing vehicle decreases in size thus reducing the suction effect, hence the higher drag. From the above two figures it can inferred that after 12m the direction of vortex just in front of the car changes from anti-clockwise to clockwise.

This reduces drag or is called favorable slipstreaming. The reverse case occur after 20m. The clockwise rotating vortex which was earlier present just in front of the sedan now does not favor the automobile. The effect of the vortex diminishes after 20m and it does not affect the drag forces on the car. Thus the drag force again starts increasing. This is evident from vorticity vector given in Fig 14.

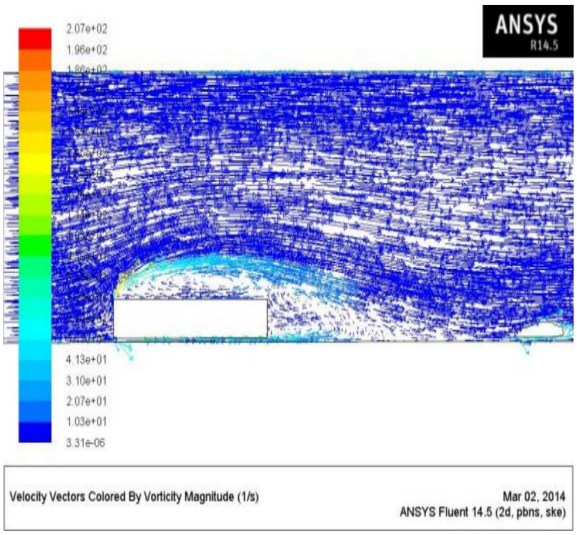


Fig 14: Vorticity vector at distance of 24 m (>20m) at speed 80kmph

When the car moves out of the vortex produced by the truck, slipstreaming effect is negligible. i.e. for all three speeds the vortices produced by the truck diminishes in strength after distance of 20m.

It is therefore clear that the slipstreaming effect is more pronounced at smaller distance of separation. From the Fig 11, it can be inferred that minimum drag occurs at 4m. But on considering safety factor it is not advisable at high speeds. So the optimum slipstream distance is taken as 20m which corresponds to second minima.

Further studies have been performed to understand the influence of slipstreaming at various speeds. The distance has been maintained at 20m, which is the critical distance, and the speed of the vehicles has been varied. The result is shown in Fig 15.

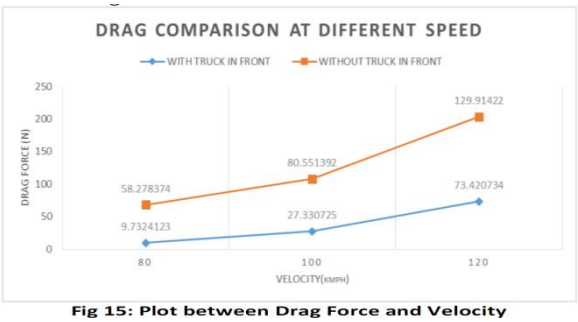


Fig 15: Plot between Drag Force and Velocity

From the graph we can infer that the slipstreaming is more effective at higher velocities of travel.

CONCLUSION

Based on the investigation performed it has been determined that slipstreaming is effective at higher velocities of the vehicles under consideration. The optimum or critical slipstreaming distance was found out. It has been observed that as the speed increases the magnitude of drag reduction increases at critical distance. From the results obtained and taking into account the practical scenario, we can conclude that a distance of 20m between vehicles is optimum for the trailing vehicle as far as drag reduction is considered, which in turn will improve vehicle performance. The study conducted was 2-D analysis, therefore there is scope for further study incorporating unsteady effects of wake, inertia and viscous effect of air flow. The study can be further extended to understand the effect of drag in convoy system, i.e. more than 2 vehicles moving together maintaining distance between them. Also 3-D analysis may be carried out to get improved accuracy.

REFERENCES

[1] Ahmed S.R., (1981), "Wake structure of typical automobile shapes", Trans. ASME, J. Fluids Eng. 103, 162-169.

[2] Gokul Krishnan Rajamani, (2006), "CFD Analysis of Air Flow Interactions in Vehicle Platoons", School of Aerospace, Mechanical and Manufacturing Engineering, RMIT University, August 2006.

[3] Han T., (1989), "Computational analysis of three-dimensional turbulent flow around a bluff body in ground proximity", AIAA J. 27(9) 1213- 1219.

[4] Martin Bjerling, (2011), "Large Eddy Simulations on a Vehicle Platoon During In- Line Oscillations", Bachelor's thesis 2011:37, Chalmers University Of Technology Gothenburg, Sweden 2011.

[5] Sanisa K. and Davidson L., (2003). "Numerical Study of the Flow around a Bus-Shaped Body." Transaction Journal of the ASME.Vol.125.

[6] Shahrin Hisham Amirnordin et al., (2010). "Numerical Analysis on the Effects of Air Flow in the Wake of a Large Vehicle on Trailing a Passenger Car", Proceedings of MUCET 2010.

★★★