International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 4 Issue 09, September-2015

CFD Analysis of Air Drag Breaking System

Vijaykumar P. Patil
Department of Mechanical Engineering,
Sinhgad Academy of Engineering, Kondhwa, 411048
Pune, India

Prof. P. P. Hujare
Department of Mechanical Engineering .
Sinhgad Academy of Engineering, Kondhwa, 411048
Pune, India

***Abstract*— The main aim of this paper is to investigate of aerodynamics of square back care and development of drag estimation of the vehicle body. Aerodynamic drag was considered for combining it with a breaking force to increase breaking efficiency. Estimation of the drag force on vehicle body is carried out with the arrangement of the Drag breaking system on the vehicle's front roof. In this paper, the technique used to estimate the drag is by using the Computational Fluid Dynamics (CFD) analysis to find the value of aerodynamic drag in terms of drag forces and drag coefficient. The result shows that contour and trajectory plot also used to analyze the characteristics of streamlines flow or boundary layer that occurs on the body of this model. The result obtained from CFD has been compared with the experimental investigation on clay model of reduced scale 1:16.**

Keywords—CFD, Air drag breaking, Wind tunnel.

I. INTRODUCTION

A vehicle that travels on a road is exposed to different kinds of resistance such as rolling, gravitational, acceleration, and aerodynamic drag resistance as the speed of a vehicle increases the aerodynamic drag resistance becomes more dominant amongst all the resistances. In latest car design drag is reduced to increase a car speed. Hence car is optimized by manufacturers by having rounded body corners, raked windows and hatchback. Aerodynamic drag is a significant factor in car's ability to accelerate. At very high speeds, the ratio of the engine's power output to the body's drag becomes more important than the power-to-weight ratio, which is important at lower speeds where drag is not significant. Aerodynamic drag can be used to help stop the vehicle. By combination of drag force with breaking force, the car will decelerate at a very high rate.

Drag force acting on a body is dependent on the geometry of the body, motion of the body and air in which it is travelling [1]. The general forces found are drag and lift. This can be proved from the equation (1)related to fluid mechanics.

$$F_d = C_d \frac{1}{2} \rho v^2 A \qquad (1)$$

Where, F_d is the drag force, C_d is the drag coefficient, ρ is the fluid density, V is the object velocity and A, being the frontal projected area. The external flow at upper side of the body is the best way as consideration for study in the aerodynamic field [2]. The drag coefficient, C_d, is a dimensionless parameter that describes how much drag is caused by airflow on an object.

II. NUMERICAL ANALYSIS

A. *DESIGN OF MODEL*

The vehicle selection criterion is a) vehicle availability in India. b) Availability of vehicle blueprints (necessary to create CAD model). The vehicle under study is selected on the basis of above two criteria. Also for experimental analysis, Bhavini [2] has taken 1:20 reduced scale aluminum car model. The car under study is 1:16 reduced scale clay model, is considered due to limitations of wind tunnel size. The dimensions of the full size car model are 4260mm in long, 1815mm wide, 1810mm high model used for analysis has dimension of 266mm long, 113.43mm wide and 113.12mm high. The drag breaking system has dimensions of 17.81mm long, 83mm wide and 16mm height and is prepared in CATIA. The pictures of the car model with and without drag breaking system is shown in figure (1).

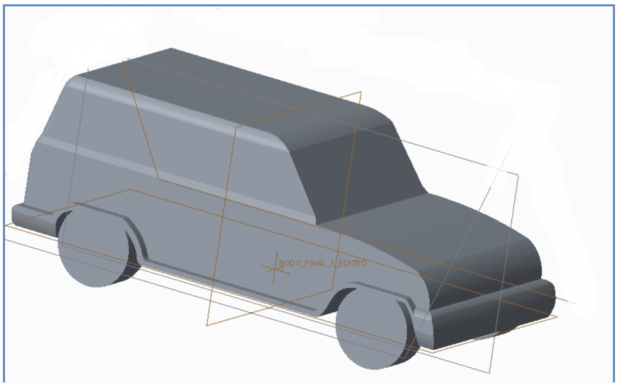

Fig 1 Square-back car model without drag breaking system

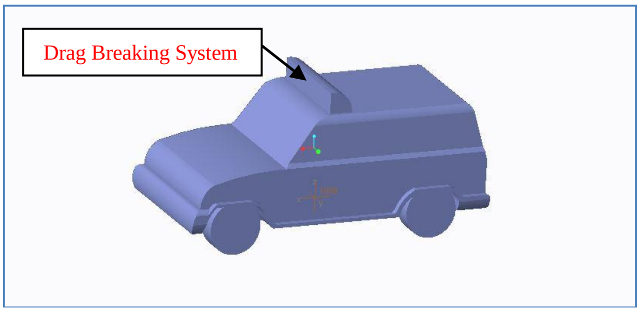

Figure 2 Square-back car model with drag breaking system.

B. *METHODOLOGY OF FEA*

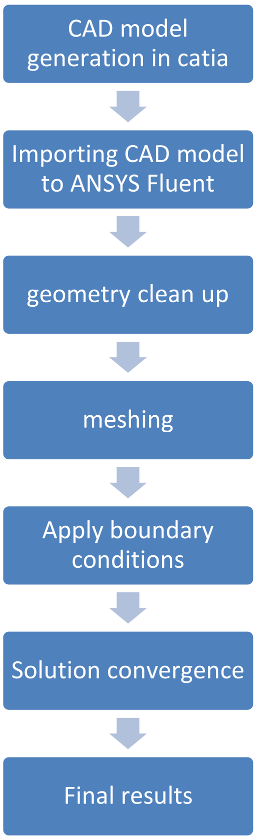

Figure 3 Methodology of FEA

First a car model is prepared in CATIA-V5 with the scale (1:16) of actual dimension. After that it is imported to the ANSYS fluent for CFD analysis. Once geometry cleanup is done, then model is further processed for meshing. Fine meshing is done near the surface of the vehicle, after that proper boundary conditions has been applied, final results obtained after solution conversions. In figure 3 above steps are given, and further explained in detail.

C. *GRID PATTERN EMPLOYED*

In this work, first of all a generic model of the square block car has been prepared in the CATIA software and this generic model imported into the ANSYS FLUENT to do the simulation of the coefficient of drag and coefficient of lift in the wind tunnel which is generated in the design module of the ANSYS FLUENT. In the present study after an extensive verification, tetrahedral grids are employed for meshing of the surface of the passenger car. To capture the wall effect as well as to save the computation time, finer grids selected near wall. The mesh employed for the car model is shown in the Figure 4.

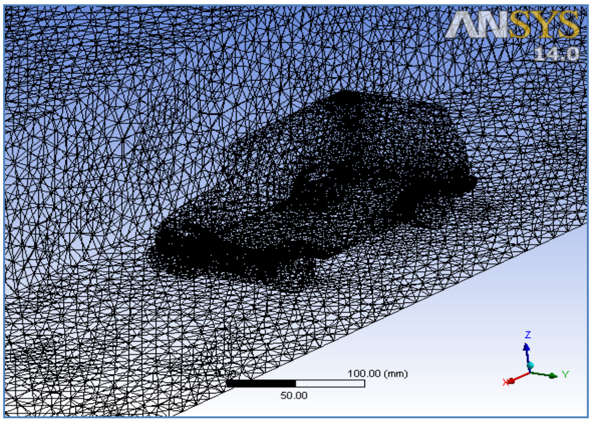

Figure 4 Meshed model of car without breaking system

D. *CFD SIMULATION AND SETUP*

The baseline model of the square back car has been designed in Solid CATIA. The size of a car model is 266mm long, 113.43mm wide and 113.12mm high. Then the model has been analyzed by using ANSYS-14.0 (FULENT) software to obtain drag coefficient and the forces.The surface mesh of square back car is generated. The tetrahedral type elements are used for surface mesh. A surface mesh of 1.5 mm size is created on the vehicle surface.

Then same procedure is repeated with the mounting Drag breaking system on the hood, and values for drag coefficient and lift coefficient is calculated. The size of the Drag breaking system is 17.81mm long, 83mm wide and 16mm height. The CFD is produced in the present simulation. Table 1, Table2, Table 3 and Table 4 shows the solver setup, viscous model and turbulence model, settings, boundary condition settings and solution controls for present simulation respectively. In present simulation, the following assumptions are considered: 1) Steady state air flow with constant velocity at the inlet.
2) The zero degree yaw angle.
3) Constant pressure outlet.
4) No slip wall boundary conditions at the vehicle surfaces.
5) Inviscid flow wall boundary condition on the top, side walls and ground face of the virtual wind tunnel. The dimension of the wind tunnel is X=15m, Y=25m and Z=20m.

Table 1: Solver setting

CFD Simulation	3-d dp (3-D Double Precision)
Solver	Fluent
Space	3D
Formulation	Implicit
Time	Steady
Velocity Formulation	Absolute
Gradient Option	Cell-Based
Porous Formulation	Superficial Velocity

Table 2:Viscous model and Turbulent model setting

Turbulence Model	k-e(2 eq.)
k-epsilon Model	Standard
Near-Wall Treatment	Enhanced wall Function
Operating Conditions	Ambient

Table 3: Solution controls

Equations	Flow and Turbulence
Discretization	• Pressure: Standard • Momentum: Second Order Upwind • Turbulence Kinetic Energy: Second Order Upwind • Turbulence Dissipation Rate: Second Order Upwind
Monitor	Residuals & Drag Coefficient
Convergence Criterion	Continuity = 0.001 X-Velocity = 0.001 Y-Velocity = 0.001 k = 0.001 epsilon = 0.001 C_d=0.001 Cl=0.001

Table 4: Boundary condition setting

Boundary Conditions		
Velocity Inlet	Magnitude (Measured normal to Boundary	20.9 m⁄s (constant)
	Turbulence Specification Method	Intensity and Viscosity Ratio
	Turbulence Intensity	10.00 %
	Turbulence Viscosity Ratio	10
Pressure Outlet	Gauge Pressure magnitude	normal to boundary
	Gauge Pressure direction	Normal to boundary
	Turbulence Specification Method	Intensity and Viscosity Ratio
	Backflow Turbulence Intensity	10%
	Backflow Turbulent Viscosity Ratio	10
Wall Zones	Vehicle surface-no slip wall B/C Ground face- inviscid wall B/C Side faces -inviscid wall B/C	
Fluid Properties	Fluid Type	Air
	Density	ρ = 1.225 (kg⁄m^3)
	Kinematic viscosity	v = 1.7894×10^(-5) (kg⁄(m·s))

E . *RESULTS AND DISCUSSION*

The CFD analysis is done at maximum speed of 20.9 m/s, for both cases with and without the drag breaking system and results were procured. Contours of pressure and velocity are plotted and value of drag coefficient is taken.

Figure 5 shows the coefficients of drag on the base model. The maximum value of the coefficient of drag for velocity of air 20.9 m/s is 0.00276057.

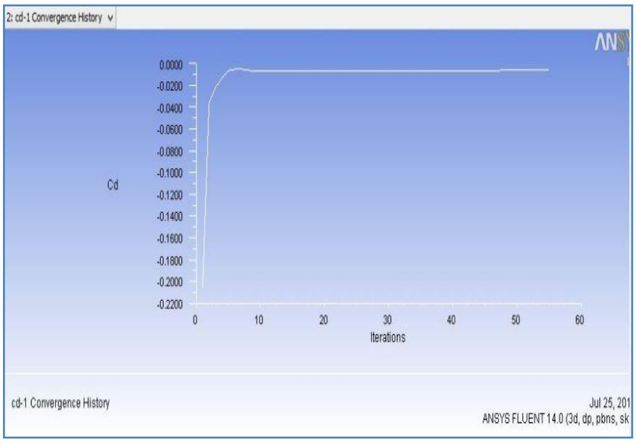

Figure 5: Coefficient of drag (C_d) without air drag breaking system

Figure 6 shows the coefficient of drag (C_d) with the air drag breaking system. Maximum value of coefficient of drag is 0.0064099.

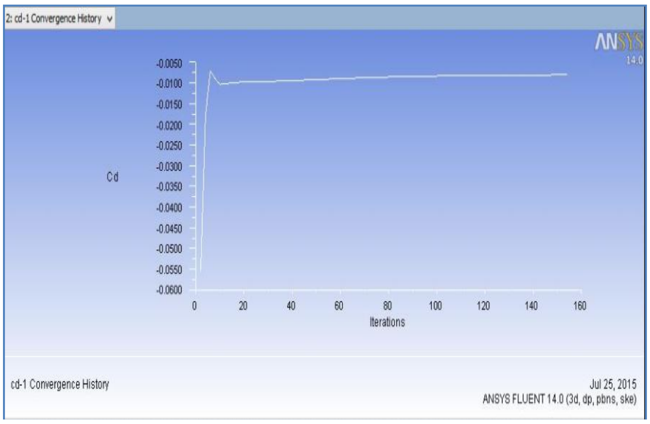

Figure 6: Coefficient of drag (C_d) with air drag breaking system

In Figure 7 contours of pressure has been shown in a car model without a drag breaking system. Variation of color on the surface of the car shows the intensity of pressure at particular sections of the car, so as it is easy to analyze an area of interest. From the picture it has seen that in the frontal region of the car there is a high pressure region generated because of a drop in velocity.

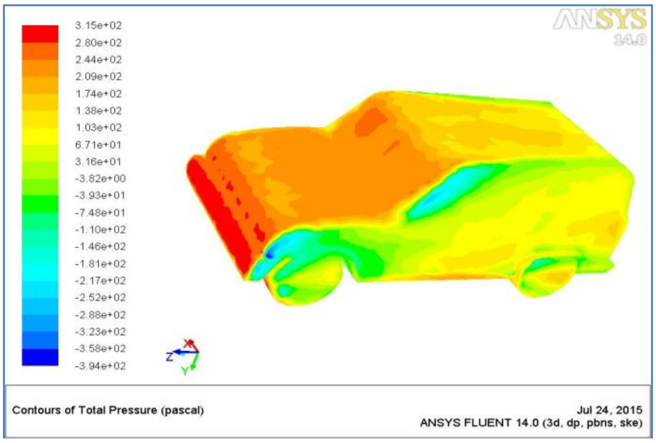

Figure 7: Total Pressure Contour on surface of car without Drag breaking system

Figure 8 illustrate, velocity streamline over the surface of the vehicle. Streamlines are a family of curves that are instantaneously tangent to the velocity vector of the flow. This shows the direction of a mass less fluid element travelled at any point in time.

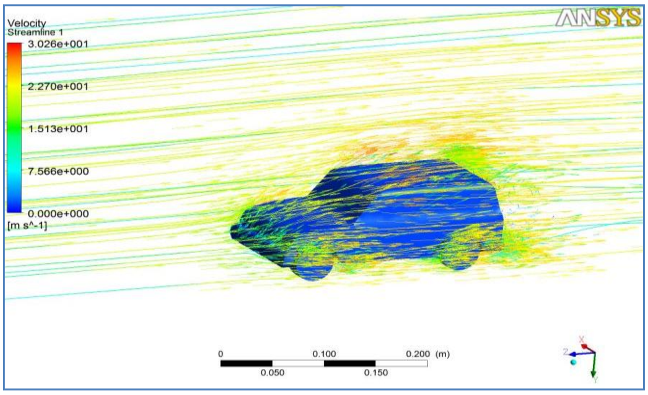

Figure 8: Velocity streamlines on surface of car without the Drag breaking system

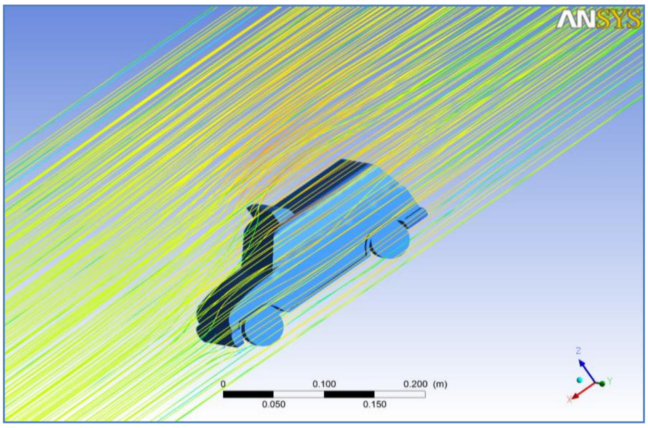

Figure 9 :Velocity streamlines on surface of car with the Drag breaking system

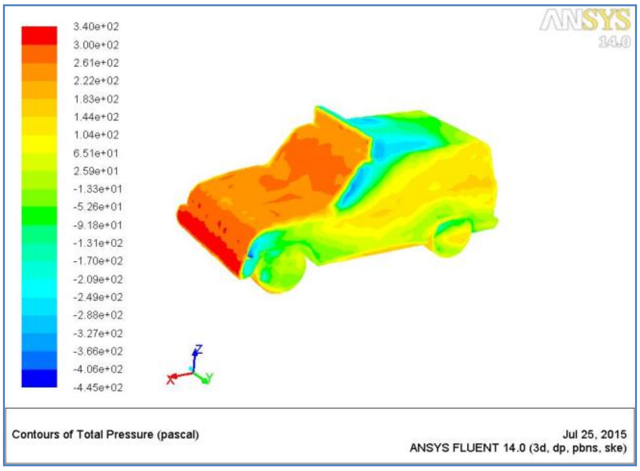

Figure 10: Total Pressure Contour on surface of car with Drag breaking system

In the case of Drag breaking system is applied to the hood of the square back car model, the coefficient of drag is 1.8. The percentage increase coefficient of drag is 132.19%. Hence Drag force on passenger car is increased as proportional to the Drag coefficient. The comparative results between the baseline car and car with the air drag breaking system are shown in Table 5

Table 5: Comparison of drag coefficient of the baseline car with a model attached Drag breaking system

Configurations	Drag Coefficient	% Coefficient of Drag Increase from baseline
Baseline	0.00276057	0
With drag, breaking system	0.0064099	132.194800

III. EXPERIMENTAL ANALYSIS

A. EXPERIMENTAL SET UP

The experimental setup using subsonic wind tunnel is shown in Figure 11

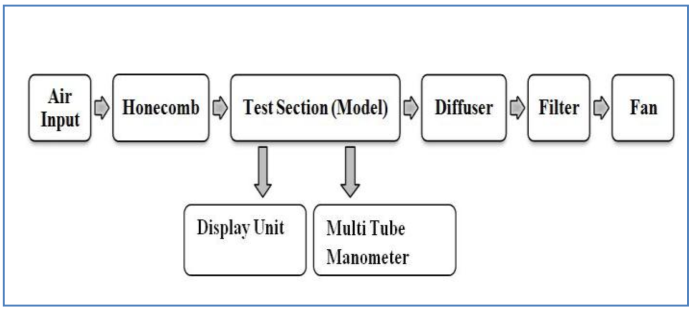

Figure 11 :Schematic diagram of experimental setup

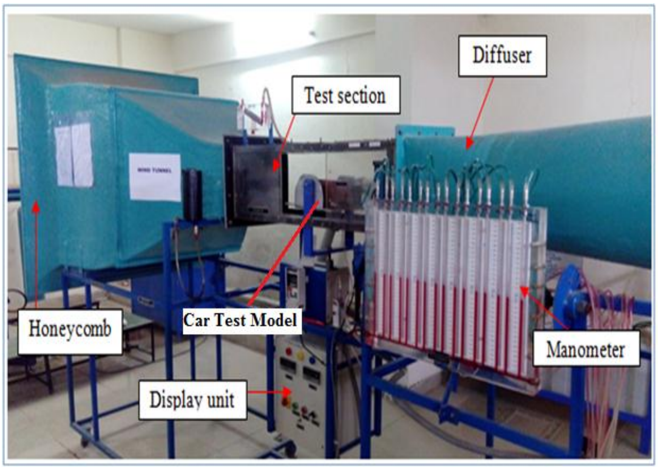

Figure 12 : Actual pictures of experimental setup

For an experimental analysis first step is to prepare a car model on which drag force is to be measured. A clay model of reduced scale (1:16) is prepared with clay modeling tools.

B. *EXPERIMENTATION PROCEDURE*

Once the model is fixed in the test section, seven connections are made to the multi tube manometer to measure pressure across the surface of the vehicle. For each connection different numbers are given to identify locations of the pressure tapping. The figure 13 show the baseline model.

Figure 13 Experimental car model for test

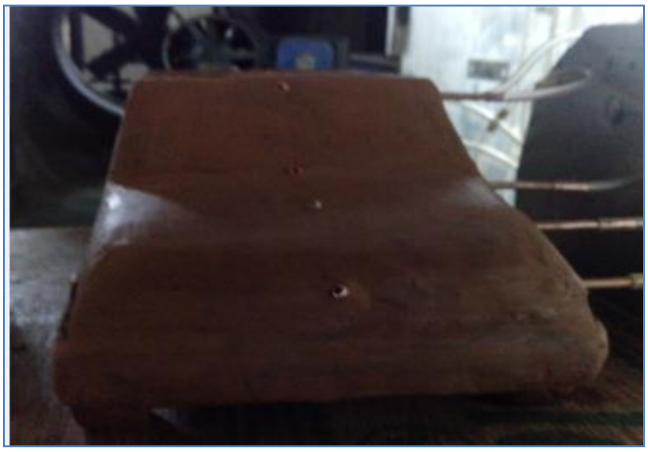

Figure 14: Car model mounted in the wind tunnel and connections are made to the manometer

Once setup is ready then perform the experimentation. The wind tunnel is started, in that speed of the wind is directly controlled by a Knob and it is directly displayed on the digital display. At the beginning velocity of air is maintained at 10 m/s velocities, and reading of the manometer is noted down. Then the speed is increased gradually and manometer readings are taken for particular velocity. The same procedure is repeated and readings have been taken at seven points for different air velocity ranging from 10 m/s to 20 m/s.

After taking the results in terms of height of water column in the manometer, experimental setup of the car with the drag breaking system is mounted in a wind tunnel. In this case one extra tube is attached to the breaking system to measure pressure at that point; now the total number of connections to the manometer is 8. Air velocity in wind tunnel increased gradually from 10 m/s to 20 m/s and readings has been taken for ten different velocities at 8 different points across the car body. The setup is shown in Figure 15

Figure 15: Experimental setup of car model with a drag breaking system

There are two methods adopted for experimentation, one relies on measurement of pressure in the effective domain upstream and downstream of car, and the other relies on pressure distribution along centerline, over car profile [7]. The method adopted for the calculation of the drag force is given below.

$$\text{Drag force } (F_d) = C_d \frac{1}{2} \rho v^2 A \qquad (2)$$

Where, P = Pressure at a particular point.
A = Car frontal area (projected area)
θ = Angle between relative velocity and normal pressure.

The car frontal area is $0.00910234 m^2$ and when the drag, breaking system is applied it becomes $0.01043034 m^2$. In this report drag force is calculated by equation (2).
With equation (2) drag force is calculated for all the values of velocities with and without drag breaking system is done. The comparison between drag forces by two settings has been done; for comparison, it is clear that model with air drag, breaking system is giving more drag force than baseline model.

C. *OUTPUT CURVES*

Figure 16 shows a graph of velocity vs drag force for cars without a drag breaking system. From this graph it can seen that ,as the speed of vehicle increases, there is increase in drag force.

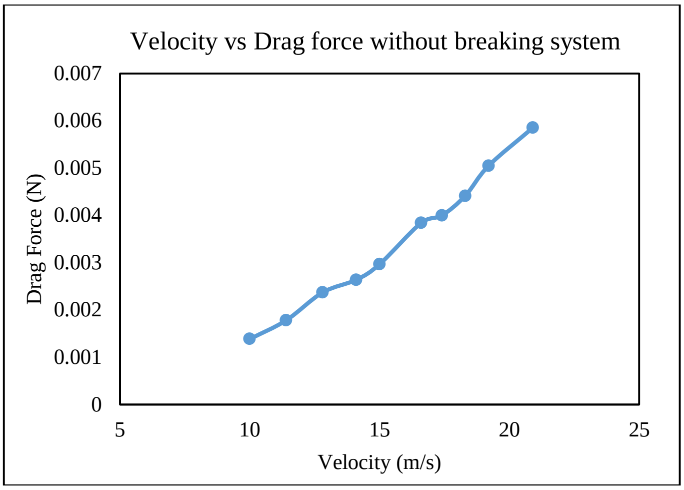

Figure 16: Velocity v's Drag force without breaking system

Similarly the graph is drawn for model with the Drag breaking system, which is shown in the figure 17.Also comparison result of both system are shown in figure 18.

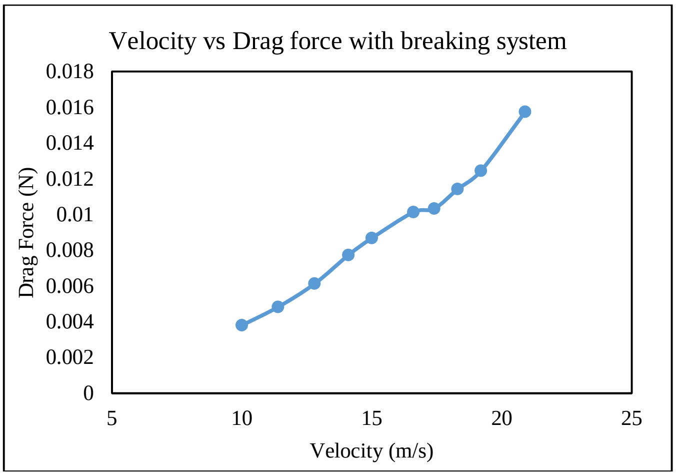

Figure 17 :Velocity Vs Drag force with breaking system

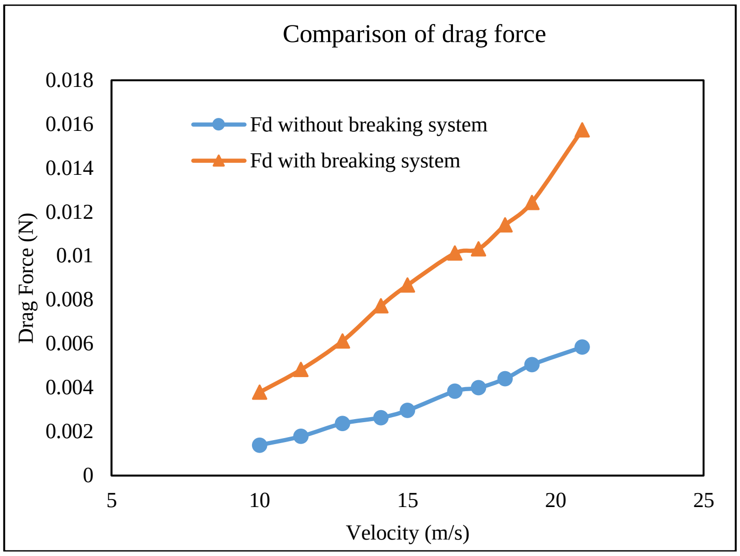

Figure 18 Comparison of drag forces with and without breaking system.

From figure 18 it can seen that, at the same velocity there is a considerable amount of increase in drag force when the drag breaking system is attached. From observations, it is clear that the objective of use of drag force for breaking purpose is achieved.

With the help of results taken for the pressure the drag forces for models with and without breaking system have been calculated, for velocities given in Table 6, and comparison is made. The Table 6 shows a comparison of drags forces.

Table 6 Drag forces at different velocities with and without drag breaking system.

Velocity m/s	F_d without breaking system	Fed with drag, breaking system
10	0.001385	0.003794
11.4	0.00178	0.004826
12.8	0.002368	0.006123
14.1	0.002637	0.007726
15	0.002966	0.008674
16.6	0.003839	0.010127
17.4	0.003996	0.010319
18.3	0.004412	0.011415
19.2	0.005047	0.01244
20.9	0.005849	0.015741

IV. RESULT AND DISCUSSION

To investigate the effect of air drag after application of the air drag breaking system, the experiment is conducted in a wind tunnel with different velocities. The drag coefficient (C_d) is calculated with respect to the velocities. The numerical and experimental results for drag coefficient are shown in Table 7. The numerical modelling results are validated with the experimental investigation with error less than 14%.

Table 7: Comparison between numerical and experimental results

		Numerical	Experimental	% Error
The results are validated for the velocity of 20.9 m/s	C_d Without breaking system	0.00276057	0.002401761	13
	C_d with breaking system	0.0064099	0.00564072	12

From Table 5 it is clear that there is a 132.19 % increase in drag force after application of the drag breaking system.

V. CONCLUSION

Computational fluid dynamics (CFD) simulations of the steady flow field around square-back car models with and without drag, breaking system were presented and compared the simulated data to each other. The ANSYS-14.0 Fluent with the k-e steady model is used for the simulations of aerodynamics. In this analysis, the coefficient of drag has been increased 30.5% and coefficient of lift has increased 5%. Hence, the Drag breaking system is an effective tool to increase the drag force on the vehicle. The effects of different aerodynamic add-on devices on flow and its structure over a square back car may be analyzed using CFD approach.

CFD approach is the better way of the future in promising faster turnaround simulation time with cheaper running cost at the same time, it offers superior capability than the experimental approach in terms of post processing of data and graphical representation of flow analysis. The analysis shows aerodynamics drag in term of drag forces or drag coefficient proportionally increased with the square of velocity in the square back car body which has a frontal area of $0.00910234 m^2$.The objective is to increase aerodynamic drag acting on the vehicle and thus improve the breaking efficiency of a passenger car. Hence, the drag force can be increased by using add on devices on the vehicle and breaking efficiency of the car can be increased.

Here two different strategies adopted for experimentation, one relies on measurement of pressure in the effective domain upstream and downstream of car, and the other relies on pressure distribution along centerline, over car profile. By numerical analysis it is observed that the increase percentage of drag coefficient due to air drag breaking system is 132.194800 %.

It is observed that the results obtained by numerical analysis and experimental analysis are well corroborated. The air drag, breaking system is a new concept which has never been applied before and it is found that due to the application of air drag breaking, the amount of breaking effort is reduced.

REFERENCES

[1] Akshay Parab, Ammar Sakarwala, Bhushan Paste, Vaibhav Patil Aerodynamic Analysis of A Car Model Using Fluent-Ansys14.5.International Journal on Recent Technologies in Mechanical and Electrical Engineering (IJRMEE) ISSN: 2349-7947, 1 (2014), pp 7-13

[2] Bhavini Bijlani1, Dr. Pravin P. Rathod2, Prof. Arvind S.Sorthiya3Experimental and Computational Drag Analysis of Sedan and Square-Back Car. International Journal of Advanced Engineering Technology E-ISSN 0976-3945IJAET/Vol. IV/ Issue II/April-June, 2013/63-65

[3] D. Ramasamy, K. Kadirgama, A. K. Amirruddin and M. Y. TaibA Vehicle Body Drag Analysis Using Computational Fluid Dynamics. National Conference in Mechanical Engineering Research and Postgraduate Students ISBN: 978-967-5080-9501 (CD ROM)

[4] R. B. Sharma1, Ram Bansal, CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction. IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 7, Issue 5 (Jul. - Aug. 2013), PP 28-35

[5] Jaspinder Singh1, Jagjit Singh Randhawa2 CFD Analysis of Aerodynamic Drag Reduction of Automobile Car - A Review International Journal of Science and Research (IJSR) ISSN (Online): 2319-7064 Paper ID: 02014156 Volume 3 Issue 6, June 2014

[6]Christoffer Håkansson, Malin J. Lenngren CFD Analysis of Aerodynamic Trailer Devices ForDrag Reduction of Heavy Duty Trucks. Master's Thesis in the Master's programme Automotive Engineering CHALMERS UNIVERSITY OF TECHNOLOGY Göteborg, Sweden 2010 Master's Thesis 2010:10

[7] Manaldesa, S.A. Channiwala and H.J.Nagarsheta, A Comparative Assessment of two Experimental Methods for Aerodynamic Performance Evaluation of a Car. Journal of scientific & Industrial Research Vol.67, July 2008, pp. 518-522

[8] JasonLeuschen, Kevin R. Cooper Full-Scale Wind Tunnel Tests of Production and Prototype, Second-Generation Aerodynamic Drag-Reducing Devices for Tractor-Trailers. 06CV-222 Copyright © 2006 SAE International

[9] Ashfaque Ansari1, RanaManoj Mourya2 Drag Force Analysis of Car by Using Low Speed Wind Tunnel. International Journal of Engineering Research and Reviews ISSN 2348-697X (Online) Vol. 2, Issue 4, pp: (144-149), October - December 2014,

[10] Shrish Chandra Duve, Mahendra Kumar Agrawal, An Experimental Study of Pressure Coefficient and Flow Using Sub Sonic Wind Tunnel The Case of A Circular Cylinder, International Journal of Emerging Technology and Advanced Engineering Volume 4, Issue 2, February 2014

[11]Masaru Koike, Tsunehisa Nagayoshi, Naoki Hamamoto, Research on Aerodynamic Drag Reduction by Vortex Generators. Mitsubishi motors TECHNICAL REVIEW 2004 no.16

[12]Volodymyr Sirenko, Roman Pavlovs'ky, Upendra S. Rohatgi, Methods of Reducing Vehicle Aerodynamic Drag, BNNLL--998083094-200-122-0C1P2 Puerto Rico, USA July 8-12, 2012