*International Journal on Applications in Mechanical and Production Engineering*

*Volume 4: Issue 1: March 2018, pp 15-18. www.aetsjournal.com* ISSN (Online) : 2395-3500

# Improving Breaking Effectiveness: An All-Inclusive CFD Examination of Air Drag in Breaking Mechanisms

**Kumar**
Selection grade lecturer
Department of Mechanical Engineering
MEI Evening Polytechnic, Rajaji Nagar, Bangalore 560010
kumar.gavi@gmail.com

**Hucheerappa H**
Selection grade lecturer
Department of Mechanical Engineering
MEI Evening Polytechnic, Rajaji Nagar, Bangalore 560010
hucheer_kwt@yahoo.com

Abstract - *Researching the aerodynamics of a square-backed car and creating a drag estimate for its body are the main goals of this project. The goal of taking aerodynamic drag into account is to improve braking efficiency by combining it with braking forces. To calculate the amount of drag acting on the vehicle's body, a Drag Breaking System mounted on the vehicle's front roof is used. The method for estimating drag is computational fluid dynamics (CFD) analysis, which yields values for the drag forces and the drag coefficients. Contour and trajectory charts are part of the results, which shed light on the body of the model's response to boundary layer effects and streamline flow. Experimental investigations on a reduced-scale clay model are compared with computational fluid dynamics (CFD) results.*

Keywords: *computational fluid dynamics (CFD)*

## I. Introduction

The forces of rolling, gravity, acceleration, and aerodynamic drag are just a few that a vehicle must overcome as it travels down a road. Among these forces, aerodynamic drag resistance grows more and more dominant as the vehicle's speed increases.

In order to increase speed, modern automobiles have rounded body corners, raked windows, and hatchbacks as an optimisation of vehicle forms for reducing drag. When travelling at high speeds, aerodynamic drag has a much greater impact on acceleration than the power-to-weight ratio, which is more important at lower speeds. Vehicles use aerodynamic drag for both acceleration and slowing, according to the auto industry. It is possible to quickly slow down the vehicle by combining the forces of drag and braking. A body's drag force is affected by its geometry, its speed, and the air it's moving through. Equation (1) shows that the two primary forces at work here are drag and lift. Here, Fd stands for drag force, Cd for drag coefficient, ρ for fluid density, V for object velocity, and A for frontal projected area. It is vital for aerodynamic studies to examine the external flow on the upper side of the body. As a dimensionless parameter, the drag coefficient (Cd) describes the effect of airflow on an object, providing information on the drag caused by the object's size and shape when it is in motion.

## II. Design Model

First, the vehicle must be available in India; second, the blueprints, which are necessary to build a CAD model, must be readily available. These two factors determine the vehicle selection process.

All of these requirements are met by the selected test vehicle. Also, because the wind tunnel isn't big enough, we use a 1:16 scale clay replica of the vehicle for testing.

The full-size vehicle has measurements of 4260 mm in length, 1815 mm in width, and 1810 mm in height; this scale model, on the other hand, is 266 mm long, 113.43 mm wide, and 113.12 metres tall. Due to space constraints in the wind tunnel, a lower size of 1:16 was used. To set the record straight, Bhavini [2] conducted their experimental research using a 1:20 scale model of an aluminium automobile. The CATIA-designed drag-breaking system is 17.81 mm long, 83 mm wide, and 16 mm high. Figure (1) shows diagrams of the vehicle model with and without the drag-breaking device. The first step is to create a 1:16 scale model of the vehicle in CATIA-V5. Afterwards, the model is loaded into ANSYS Fluent to do CFD analysis. After the geometry cleanup is finished, the model is meshed, with an emphasis on making a fine mesh close to the vehicle's surface. When the meshing is complete, the solution is allowed to converge before applying the necessary boundary conditions and finally, the results are produced. These steps are illustrated in Figure 3 and explained in full below.

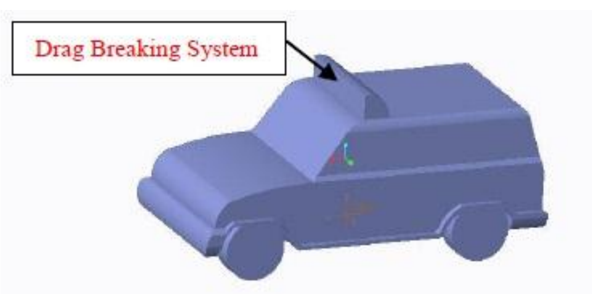


Figure 1. Square Back Breaking model

## . III. Grid Pattern

Building a detailed model of a square block vehicle in CATIA is the first step in our investigation. In order to simulate drag and lift coefficients in a wind tunnel, the generic model is imported into ANSYS FLUENT and then run through the program's design module.

After careful confirmation, the surface of the passenger automobile is mesh-researched using tetrahedral grids. By placing finer grids strategically near the wall, we may optimise calculation time and account for the wall effect. Figure 4 shows a visual representation of the car model's mesh configuration.


Figure 2. Methodology

## IV. CFD Simulation

The 266 mm length, 113.43 mm width, and 113.12 mm height initial model of the square back automobile is built in Solid CATIA. The model is then evaluated by computing the drag coefficient and forces using the ANSYS-14.0 (FLUENT) programme. For the vehicle's exterior, a 1.5 mm tetrahedral surface mesh is produced.

*Volume 4: Issue 1: March 2018, pp 15-18. www.aetsjournal.com* ISSN (Online) : 2395-3500

Using the hood's Drag Breaking System to calculate lift and drag coefficients is much the same as the previous step. Here we offer the computational fluid dynamics (CFD) results of this simulation, which detail the configuration of the solver, the viscous and turbulence models, the parameters, the boundary condition settings, and the solution controls.

The following conditions are assumed: a virtual wind tunnel with dimensions X=15m, Y=25m, and Z=20m; a steady-state airflow with a constant velocity at the inlet; a zero-degree yaw angle; a constant pressure outlet; no-slip wall boundary conditions on the vehicle surfaces; and inviscid flow wall boundary conditions on the top, side walls, and ground face.

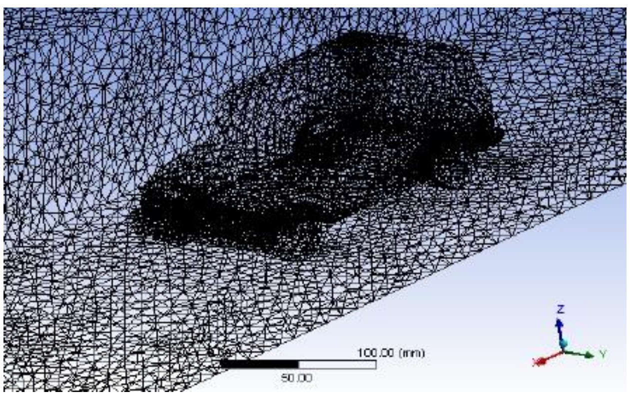


Figure 3. Grid Pattern

## V. Discussion

The model is fastened in the test portion before the multi-tube manometer is connected in seven different ways to measure pressure all over the vehicle. Each connection's location is identified by a unique number. Figure 13 displays the baseline model. As soon as everything is in place, testing may start. A digital monitor shows the current velocity as soon as the wind tunnel is turned on, and a knob adjusts the wind speed. Manometer data are obtained at an initial air velocity of 10 m/s in the experiment. Readings are taken at velocities ranging from 10 m/s to 20 m/s when the speed is progressively raised. The experimental setup including the vehicle and the wind tunnel drag-breaking technology follows the same protocol.

The breaking system has an additional tube connected to it so that pressure may be measured at that point; this makes eight connections to the manometer in total. Readings are recorded at eight distinct locations at ten different velocities as the air velocity is progressively raised from 10 m/s to 20 m/s. After using the drag-breaking technology, the car's frontal area increases from 0.00910234m² to 0.01043034m².

The drag force is determined in this report by utilising equation (2). For all values of velocity, with or without the drag-breaking device, the drag force is calculated. In the case of the model equipped with the drag-breaking mechanism, the drag force is significantly larger than that of the baseline model when comparing the two settings.

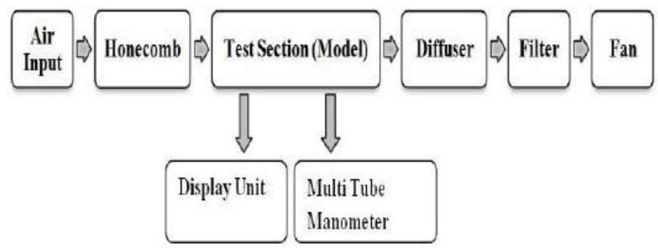


Figure 4: Experimental Setup

## VI. Conclusion

For light-duty cars, a high-efficiency gasoline-fueled CCI engine is a major leap over SIDI engines. With no major influence on fuel-refining capability and without posing major issues related to NOx and PM emission management, CCI engines have the potential to match or even exceed the efficiency of diesel-fueled CIDI engines. Furthermore, because CCI engines most likely employ lower-pressure fuel injection, they should be more affordable than CIDI engines. The way that CCI burns may also make it possible to use emission control technologies that use fewer expensive and rare precious metals. Furthermore, the creation of the diesel-powered CCI engine is noteworthy since it provides a different approach for heavy-duty vehicles.

## Reference

[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