A Review paper on Aerodynamic Drag Reduction and CFD Analysis of Vehicles

Subhasis Sarkar[1], Kunj Thummar[2], Neel Shah[3], Vishal Vagrecha[4]

[1]*Assistant Professor, Department of Mechanical Engineering, Babaria Institute of Technology, Gujarat, India*
[2,3,4]*Student, Department of Mechanical Engineering, Babaria Institute of Technology, Gujarat, India*

Abstract - *Aerodynamics is the most important factor when it comes to resistive forces acting on the vehicle. It comes into the picture when a vehicle is moving in a fluid medium. There are numerous factors such as lift, side force and drag which are responsible for this resistance. Reducing the aerodynamic drag will not only open the doors for higher top speed but will also reduce the overall fuel consumption of the vehicle and increase comfortability. These above factors are very vital when it comes to passenger cars. These factors also determine the popularity and set the base for marketing strategies for a particular passenger car. Hence, various researchers are constantly trying to optimize car design features due to above mentioned reasons. In the following review paper you will find various researches which have already been done in order to reduce the drag in various segments of vehicles.*

Key Words: Aerodynamic drag, Computational fluid Dynamics (CFD) analysis, Fuel economy, Wind tunnel

1. INTRODUCTION

There are different types of forces acting on a vehicle when it is in motion such as drag force and down force. Drag force being the more prominent one is more responsible for increased fuel consumption and lower top speed of a vehicle. There are various types of drag forces acting on a vehicle namely: Parasitic drag, lift, induced drag and wave drag. Parasitic drag is further sub divided into form, skin friction and interference drags. These individual drags are very difficult to calculate and hence most people are concerned in finding the overall drag coefficient of a vehicle. This can be found out in wind tunnels by making numerous scaled models of vehicles to be tested. The basic formula for overall drag is given by:

$$D = \frac{\rho}{2} C_d A V^2$$

Where,

C_d = Coefficient of Drag

A = Frontal area

V = Relative velocity of the object w.r.t. fluid medium

P = Density of air

Here we can observe that density of air cannot be varied and is constant. The one thing where major changes can be made is frontal area. Therefore, optimizing the frontal area i.e. modifying the car design can help solving the problem of increased drag. Car design can be changed by the use of numerous aerodynamic aids such as Air dams, spoilers, vortex generators, etc.

2. LITERATURE REVIEW

R.H. Heald. (1933)[1] investigated four models of car for its drag coefficient and compared it with the one which was 10 years earlier. He found out that elimination of fenders and other projections together with pronounced fairing of body of one model reduced the drag coefficient quite significantly. An additional decrease in value of drag coefficient was observed by eliminating the windshield and fairing the whole body of car so as to resemble a thick air plane wing section.

Shobit Sengar Et al. (2014)[2] determined forces acting on three different segments of a vehicle, the Hindustan Ambassador, Lamborghini Aventador LP 700-4 and an F1 car, by testing their models in a Wind tunnel. A comparison is done between the three models for the best aerodynamic features. The scaled models are tested under different wind conditions in. It was found that the F1 car is the most aerodynamic amongst the three followed by the Aventador and then ambassador. The former two has this result due to its low slung body which results in lower ground clearance. Also the linings of the coupe help in channelizing the air when the vehicle is in motion which leads the air to the rear end where spoilers are provided which provides additional stability at high speeds.

Abdellah Ait Moussa Et al. (2015)[3] worked on reduction of Aerodynamic Drag in generic trucks using geometrically optimized rear cabin bumps. They used a 1/10th scaled half model of a generic truck and added three equally spaced bumps on the top of the cabin surface. Thereafter they used Taguchi or Orthogonal array optimization method to study the effect of these bumps on drag. Next, they tested Solid works model with and without bumps in ANSYS workbench and plotted a curve for pressure distribution over the cabin. They concluded that the optimal geometrical parameters of the bumps Should be as follows for maximum drag reduction, i.e. here 9.83%: (W/H0) = 0.088, (L1/H0) = 0.334, (L2/H0) = 0.078 and (h/H0) = 0.062, where W, H0, H1 and h are:

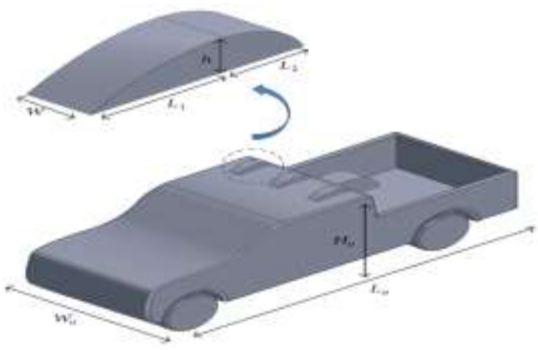

Fig -1: Generic truck test model

Taherkhani AR Et Al. (2015)[4] in this paper experimental and computational investigation into the aerodynamics of emergency response vehicles has been done and it focuses on reducing the additional drag that results from the customary practice of adding light-bars onto the vehicles' roofs. They found that reducing the fuel consumption of the YAST's ambulances during its fleet operations by 5% would save £350,000 annually and reduce the associated carbon emissions by 250 tons of CO_2, savings which could be extended throughout the UK's NHS national fleet.

Jeff Howell Et al. (2013)[5] tested a bluff body in a wind tunnel for finding the drag coefficient and lift coefficient. The model approximately replicated a quarter scaled hatchback cars. They varied the lengths from 0.075m to 0.225m and angle of taper from 5˚ to 25˚ for finding out the value of drag in different conditions. The results were plotted as below:

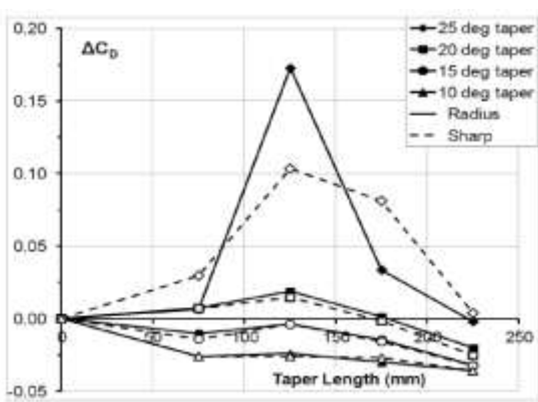

Chart -1: Effect of taper length and angle on the drag coefficient.

Yingchao Zhang Et Al. (2009)[6] the details of the virtual wind tunnel test simulation were narrated in this paper. Applying the virtual wind tunnel test aerodynamic drag coefficient, velocity contour and pressure distribution were got. Some advices to reduce aerodynamic drag of the design car were put forward. It was found that it is a simple, effective, convenient and fast way to do aerodynamic numerical simulation based CFD in the process of car styling.

Abdulkareem SH. Mahdi – Obaidi Et al. (2014)[7] tested an open wheel race car made by students of Taylor's University in a wind tunnel and in ANSYS fluent. They studied the effect of Radiator air channel in drag optimization and compared both experimental and numerical results. They found out that increasing the angle of tilt of radiator channel from 36˚ to 72.5˚ results in reduction in drag to 0.563 from 0.619. Both results agreed without much deviation in results. There was only 7.7% between both results.

J Abhinesh Et al. (2014)[8] conducted a CFD analysis of two Volvo intercity buses. Model one being the existing bus model and second being the modified one. This they did in order to reduce the aerodynamic drag and fuel consumption. After the CFD analysis they found drag reduction of about 10%. The Original model's coefficient of drag was found to be 0.8 and for the modified model was found to be 0.7.

Francesco Mariani Et al. (2012)[9] numerically tested a race car model which was made by the students of University of Perugia. Their main experiment was focused around changing the design of car nose so as to optimize the aerodynamic drag. They called the original model as "A" and modified as "B". In model B they added a front wing, modified the headrest, adopted an air extractor and added a wing on front tire. The results obtained are plotted below:

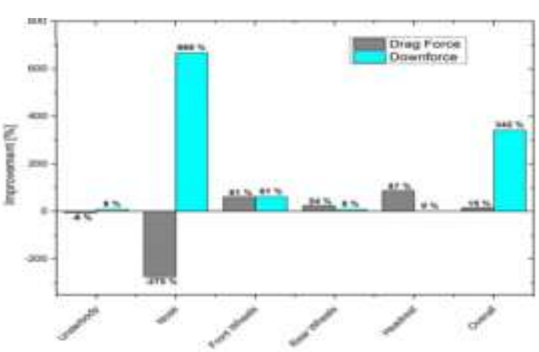

Fig -2: Percentage improvement of the forces on Model B in relation to Model A

Ashfaque Et al. (2014)[10] discussed the Drag force analysis of car by using low speed wind tunnel. In the testing they used the Pitot tube, manometer and solid object (airfoil). They calculated the velocity of fluid and drag & lift forces. They observed that design of low speed open circuit wind tunnel is somewhat different to other wind tunnel. Its diffuser is flexible. The construction of machine is low cost and Design is very easy. It's using materials easily available in the market. The machine is useful to educational and research purpose.

Keisuke Nisugi Et al. (2004)[11] worked on reduction of Aerodynamic Drag for vehicle having feedback flow control. In their study they mounted a sensor (control flow nozzle) which provided the controller the information about velocity and pressure components. As per the requirement the controller drives the actuator which in turn operates the control port where blowing and suction of air takes place.The nozzle was placed in a portion of the front wind shield. Proper systametic calculation resulted in 20% drag reduction as compared to the vehicle without the feedback flow control system.

L. Anantha Raman Et al. (2016)[12] conducted a comparative study of different methods of aerodynamic drag reduction to reduce fuel consumption in vehicles. They conducted passive tests on a SUV model by extending its rear end (rear fairing), adding a rear plate (rear screen) and by adding a vortex generator (Delta wing and bump shaped). A 6.5% and 26% reduction of drag was found by installation of rear screens and rear fairing respectively. Among the vortex generators the delta wing type were foung more effective drag reducers.

Upendra S. Rohtagi Et al. (2012)[13] tested a small scale model of General Motor SUV and tested in the wind tunnel for expected wind conditions and road clearance. Two passive devices, rear screen which is plate behind the car and rear fairing where the end of the car is aerodynamically extended, were incorporated in the model and tested in the wind tunnel for different wind conditions. The conclusion was that rear screen could reduce drag up to 6.5% and rear fairing can reduce the drag by 26%. It was also mentioned that efficiency of rear screens from point of view of drag reducing equally depends on configuration, dimensions and arrangement of screens as well as on model's rear part configuration.

S.M. Rakibul Hassan Et al. (2014)[14] numerical methods test to reduce the effect of aerodynamkic drag in a racing car. The did under body modification by slicing the underboady which allows more air to be suctioned in the low pressure zon. The plot of C_d v/s Slicing angle is shown below. Fair amount of reduction in drag was experienced. Another method employed was redirecting the exhaust gases at an angle of 45˚ towards the low pressure zone behind the car to minimize the effect of negative pressure. It was observed that with decrease in the exhaust velocity the coeficient of drag also was found to reduce.

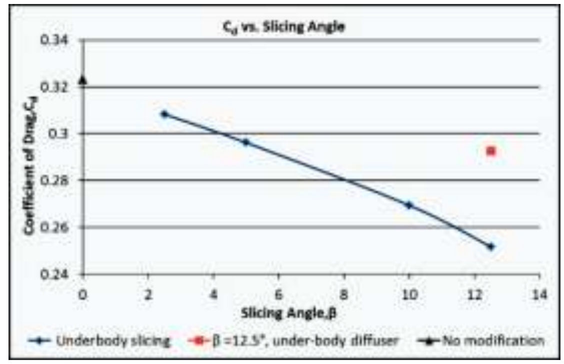

Fig -3: Coefficient of drag v/s Slicing angle

Mohd Nizam Sudin Et al. (2014)[15] reviewed the performance of active and passive flow control on the vehicle aerodynamic drag reduction is reported in this paper. The review mainly focuses on the methods employed to prevent or delay air flow separation at the rear end of vehicle. Passive methods i.e. Vortex Generator (VG), spoiler and splitter and active flow controls i.e. steady blowing, suction and air jet are among the methods are been reviewed. They found out that aerodynamic drag is responsible for 50% fuel consumption in a vehicle. It was observed that active flow control is more preferable in reducing these effects as compared to passive flow controls.

Rose McCallen Et al. (1999)[16] conducted Wind tunnel analysis of model of 1:14 Class 7 & and Class 8 heavy duty Sandia trucks to reduce their aerodynamic drag and so as to improve the fuel economy. The PIV (Particle image velocimetry) measurements were taken in the model wake. Oil film interferometry techniques (OFI) for measuring skin friction and pressure sensitive paint (PSP) measurements were also used. They found that PIV approach to calculate various parameters in the Wind tunnel can be effective in finding more precise and accurate results.

Yiping Wang Et al. (2016)[17] carried out numerical simulation on a generic vehicle to optimize the aerodynamic drag by employing a dimpled non smooth surface. Dimpled surface helps in turbulent air flow around the vehicle thereby delaying the separation point and hence obtaining a smaller wake and lesser form drag. They had used Kirging surrogate model to design the dimpled non smooth surface. The model used is shown below. The software used for CFD was ANSYS fluent. It was found that reduction in drag coefficient was obtained when air speed was below 10 ms^{-1} and a constant drag coefficient was found in the speed range above 20 ms^{-1}.

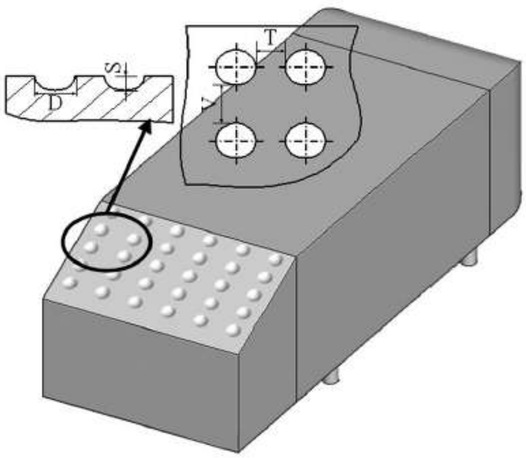

Fig -3: Ahmed body with dimples rear slant back

Pikula Boran Et al. (2011)[18] conducted Aerodynamic analysis of the exterior of a Peugeot 407 Coupe in the scale of 1:18 numerically in ANSYS fluent simulation software. They found out the pressure distribution around the car in motion. The results obtained were compared with the experimental results and found to be very similar with only little error. But they concluded that for in depth analysis such as flow around the car engine, Wind tunnel tests should be preferred over CFD.

3. CONCLUSIONS

After reading the above research papers it can be judged that aerodynamic drag is the most important factor which is responsible for Fuel consumption, power loss and top speed in a vehicle. It is also concluded that the external design features of a vehicle account highest in reducing the effects of drag in a vehicle. Every researcher above has followed the same idea of modification of external design features. Addition of vortex generators, rear screen, rear fairing, fenders, etc. are some of the remedies followed. A few of them worked on varying the rear taper angle, rear underbody angle, etc.

All the above modifications have mostly been done Race cars. A few in passengers cars and high load vehicles too. But none of the above people have tries adding front and rear spoilers in a "passenger vehicle" to reduce drag effect. Hence, there is a scope of work in this direction as well. These additions can be proved fruitful in high end passenger vehicles where cost is not a very significant factor for the buyers.

4. REFERENCES

[1] R.H. Heald, "Aerodynamic characteristics of automobile models," Part of Bureau of Standards Journal of Research, Vol 11, August 1933.

[2] Shobhit Senger and S.D. Rahul Bhardwaj, "Aerodynamic design of F1 and normal cars and their effect on performance," International Review of Applied Engineering Research. ISSN 2248-9967 Vol. 4, Number 4 (2014), pp. 363-370.

[3] Abdellah Ait Moussa, Justin Fischer, and Rohan Yadav, "Aerodynamic drag reduction for a generic truck using geometrically optimized rear cabin bumps," Hindawi Publishing Corporation Journal of Engineering Vol. 2015, Article ID 789475.

[4] Taherkhani AR, deBoer GN, Gaskell PH2, Gilkeson CA, Hewson RW, Keech A, Thompson HM and Toropov VV, "Aerodynamic drag reduction of emergency response vehicles," Advanced Automobile Engineering Vol. 4 Issue 2 - 1000122 ISSN: 2167-7670 AAE, 2015.

[5] Jeff Howell, Martin Passmore and Simon Tuplin, "Aerodynamic drag reduction on a simple car like shape with rear upper body Taper," SAE International 04/08/2013.

[6] Yingchao Zhang, Zhe Zhang, Shuanghu Luo and Jianhua Tian, "Aerodynamic numerical simulation in the process of car styling," Applied Mechanics and Materials Vol. 16-19 (2009), pp 862-865.

[7] Abdulkareem SH, Mahdi Al Obaidi and Lee Chung Sun, "Calculation and optimization of aerodynamic drag of an open wheel car," Journal of Engineering Science and Technology EURECA 2013 Special Issue August (2014) 1 – 15.

[8] J Abinesh and J Arunkumar, "CFD analysis of aerodynamic drag reduction and improve fuel economy," International Journal of Mechanical Engineering and Robotics Research ISSN 2278 – 0149, Vol. 3, No. 4, October, 2014.

[9] Francesco Mariani, Claudio Pogianni, Francesco Risi and Lorenzo Scappaticci, "Formula SAE Racing Car: Experimental and Numerical Analysis of the External Aerodynamics," 69th Conference of Italian Thermal Machines Engineering Association, ATI 2014.

[10] Ashfaque Ansari and Rana Manoj Mourya, "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, and pp: (144-149), Month: October - December 2014.

[11] Keisuke Nisugi, Toshiyuki Hayase and Atshushi Shirai, "Fundamental study of aerodynamic reduction for vehicle with feedback flow control," JSME International Journal. Series B, Vol. 47, No. 03, 2004.

[12] L. Anantha Raman and Rahul Hari H., "Methods for reducing aerodynamic drag in vehicles and thus acquiring fuel economy," Journal of advanced engineering research, ISSN: 2393-8447, Vol. 3, Issue 1, 2016, pp 26-32.

[13] Upendra S. Rohatgi, Volodymyr Sirenko and Roman Pavlovs'ky, "Methods of reducing vehicle aerodynamic drag," Presented at ASME 2012 Summer Heat transfer conference, Puerto USA, July 8-12 2012.

[14] S.M. Rakibul Hassan, Toukir Islam, Mohammad Ali and Md. Quamrul Islam, "Numerical study on aerodynamic drag reduction of racing cars," 10th International Conference on Mechanical Engineering, ICME 2013.

[15] Mohd Nizam Sudin, Mohd Azman Abdullah, Shamsul Anuar Shamsuddin, and Faiz Radza Ramli ans Musthafha Mohd Tahir, "Review of research on Aerodynamic Drag Reduction Methods," International Journal of Mechanical and Mechatronics Engineering IJMME-IJENS Vol: 14 No: 02.

[16] Rose McCallen, Fred Browand, Anthony Leonard, Mark Brady, Kambiz Salari, Walter Rutledge, James Ross, Bruce Storms and J.T. Heineck, "Progress in Reducing Aerodynamic Drag for Higher Efficiency of Heavy Duty Trucks (Class 7-8)," 1999 Society of Automotive engineers Government/Industry meeting Washington, DC. April 26-28 1999.

[17] Yiping Wang, Cheng Wu, Gangfen Tan and Yadong Deng, "Reduction in the aerodynamic drag around a generic vehicle by using a non-smooth surface," Journal of Automobile Engineering. Vol: 231(1), 130-144.

[18] Pikula Boran, Filipovic Ivan and Kepnik Goran, "Research of the external aerodynamics of the vehicle model," conference paper, May 2011.