Variation in Aerodynamic Drag and Lift by addition of a Rear Spoiler in a Passenger Car

[1]Kunj Thummar, [2]Neel Shah, [3]Vishal Vaghrecha, [4]Subhasis Sarkar
[1,2,3]Student, [4]Assistant Professor
[1,2,3,4]Mechanical Engineering Department
[1,2,3,4]Babaria Institute of Technology, Vadodara, India

Abstract: Aerodynamics plays a very vital role when a vehicle is in moving condition. It has an important role to play in case of cars, especially on passenger cars moving on highways. It is because the aerodynamic drag and lift has a direct effect on the fuel consumption of the car engine. There are various aerodynamic aids which help in reducing these effects. The objective of the current study is to find out the drag and lift coefficients of a Mitsubishi Lancer, before and after adding a rear spoiler in it. A CFD analysis has been conducted on the CAD model of the car at 4 different speeds in highway range i.e. between 75kmph and 150kmph. Results depict decrease in lift coefficient by 28.255%, which is a very significant change. Conversely, an increase in drag coefficient has been found by 5.092%. It has also been observed that regardless of the car geometry both drag and lift coefficients decrease with the increase in speed of the car.

Key words: Drag, Lift, Drag coefficient, Lift coefficient, CFD analysis, CREO, ANSYS

I. INTRODUCTION

There are different types of forces acting on a vehicle when it is in motion such as drag force and lift 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 drag. 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:

$$D = \frac{\rho}{2} C_d A V^2 \qquad \text{(i)}$$

Where,
D = Drag force, N
C_d = Coefficient of Drag
A = Frontal area, m^2
V = Relative velocity of the object w.r.t. fluid medium, ms^{-1}
P = Density of air kgm^{-3}
The concept for Lift is very similar to that of drag. Even though it is lower in magnitude as compared to the drag but has a significant effect on rideability of a vehicle. Mathematically it is expressed as:

$$L = \frac{\rho}{2} C_l A V^2 \qquad \text{(ii)}$$

Where,
L = Lift force, N
C_l = Coefficient of Lift
All other variables have the same usual meaning.
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

Abdulkareem SH. Mahdi – Obaidi Et al. (2014) [1] 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.

Abdellah Ait Moussa Et al. (2015) [2] 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%: $(W/H_0) = 0.088$, $(L_1/H_0) = 0.334$, $(L_2/H_0) = 0.078$ and $(h/H_0) = 0.062$, where W, H_0, H_1 and h are:

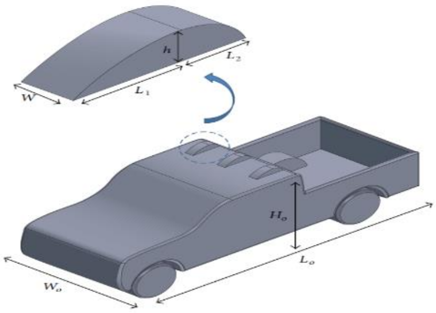


Fig. 1 Generic truck test model

Taherkhani AR Et Al. (2015) [3] 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 CO2, savings which could be extended throughout the UK's NHS national fleet.

L. Anantha Raman Et al. (2016) [4] 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.

S.M. Rakibul Hassan Et al. (2014) [5] 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.

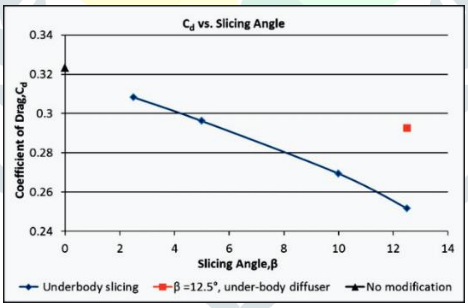


Fig. 2 Coefficient of drag v/s Slicing angle

Yiping Wang Et al. (2016) [6] 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}.

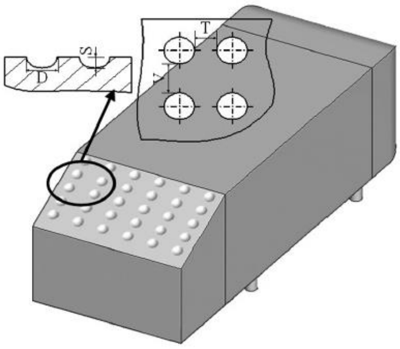


Fig. 3 Ahmed body with dimples rear slant back

3. METHODOLOGY

3.1 Vehicle selection and modelling:

There are many different geometries of cars available in the Indian automotive market. Since our study was related to the rear spoilers, the SUVs and the Hatchbacks were already eliminated, leaving with us only one option to choose i.e. a sedan. Now, there are already a few manufacturers who are using this concept of rear spoilers, such as, Suzuki Ciaz, Honda City, etc. But it was not sure that they are using the spoiler as an aesthetic purpose or any other more useful purpose. Hence, we thought to select a very similar car and add somewhat similar spoiler in it and then test it under different wind conditions. Finally, for our purpose we chose a Mitsubishi Lancer. It has a very similar geometrical construction as we desired for and also it has enough provision at the back to add a spoiler to its original geometry. We modelled our car in Creo 5.0. The cars models looked as shown below.

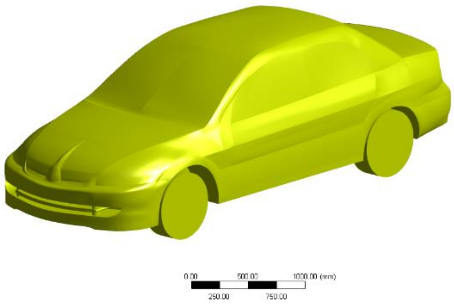


Fig. 4. Mitsubishi Lancer (original)

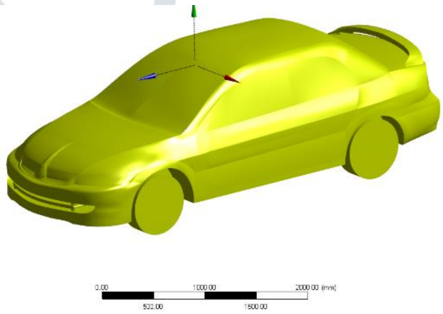


Fig. 5. Mitsubishi Lancer (with spoiler)

From the original car model, we made four eliminations as follows:

- Both front and rear wipers were not modelled,
- Side rear view mirrors were not modelled,
- The Manufacturer's logo was not included and
- Tires were not made like conventional ones (with grooves).

We made the above-mentioned eliminations due to only one reason i.e. all these parts are very intricate and has a very minute and groovy geometry. Due to this intricacy it would have been difficult to generate the mesh while performing CFD analysis. Moreover, these parts don't have much role to play in the mechanics of air flow around the vehicle. Hence, its removal will only have a little effect in our results.

3.2 CFD analysis setup:

The first step was to setup object enclosure. The dimensions of the enclosure are: Height = 4200mm, Width = 5000 mm and Length 12000mm. The enclosure for both with and without spoiler model is as follows:

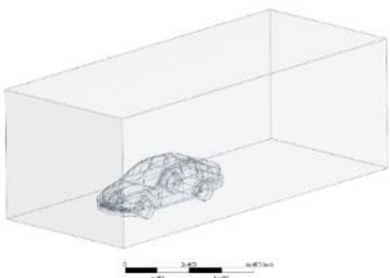


Fig. 6. Fluid domain

After creating an enclosure, the faces of the enclosure are assigned their respective parameters i.e. Inlet, opening, ground, car and spoiler. Among these grounds, car and the spoiler are considered as walls and opening as an atmosphere. All the walls are considered as smooth walls i.e. air can easily pass over the surface without any hinderance. Parameters are same for both the models.

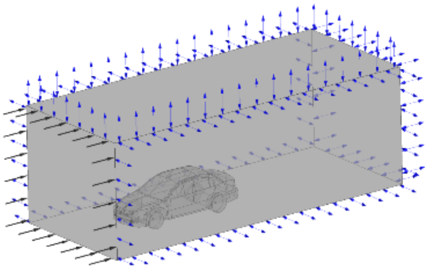


Fig. 7. Final CFX pre

After the fluid domain has been set up it is followed by meshing. Mesh generation is the practice of generating a polygonal or polyhedral mesh that approximates a geometric domain. More nodes will give us more accurate and precise results. Our meshed model is as follows n our analysis we used the tetrahedral type of mesh. Total nodes generated in our model were approximately 5 Lakhs. More nodes can be generated by increasing the fineness of the mesh. More nodes will give us more accurate and precise results. Our meshed model is as shown below:

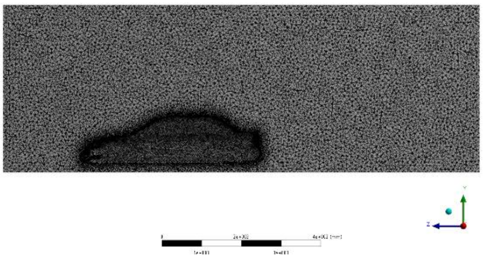


Fig. 8. Mesh model (without spoiler)

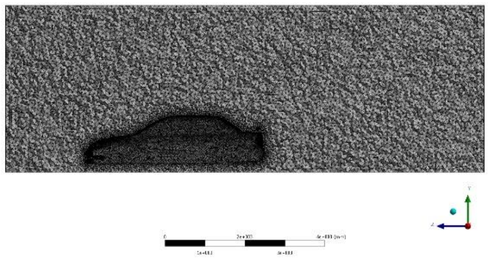


Fig. 9. Mesh model (with spoiler)

3.3 CFD analysis:

Normally, on Indian highways the speed of a vehicle ranges from 75 – 150kmph. Hence, we did CFD analysis for 4 different speeds in this range. We had considered analysis for 200 iterations or $1 * 10^{-5}$ conversion, whichever comes first. The ultimate aim of the project was to find out the drag and lift coefficient of the cars. But these coefficients are dependent and derived quantities i.e. they can't be obtained directly from any experiment. We have to find other quantities on which they depend and then calculate the required quantities. In our study we found out drag force and the lift force. Values of forces are tabulated below:

Velocity (m\s)	Without spoiler		With spoiler			
	Drag force (N)	Lift force (N)	Drag force due to car (N)	Drag force due to spoiler (N)	Lift force due to car (N)	Lift force due to spoiler (N)
20.833	176.229	83.411	171.006	14.801	69.115	-8.325
27.777	312.633	146.901	302.359	26.569	120.943	-14.801
34.722	487.97	228.94	470.243	41.81	186.192	-23.15
41.666	702.02	329.07	675.071	60.56	265.849	-33.405

Table 1. Forces obtained for both the car models

3.4 Final results (deriving C_d and C_l):

Since we have already got the values of drag force and lift force, coefficient of drag and coefficient of lift can be easily calculated from the equation (i) and (ii). Density of air in all cases is taken as 1.185 kgm^{-3} and the frontal area as calculated by the software is 1.937 m^2, for all cases. Values of C_d and C_l are tabulated below.

Velocity (ms^{-1})	With spoiler		Without spoiler		% change in C_d	% change in L_d
	Drag coefficient	Lift coefficient	Drag coefficient	Lift coefficient		
20.833	0.3538	0.16746	0.37303	0.12204	5.435 (↑)	27.123 (↓)
27.777	0.35306	0.1659	0.37146	0.11987	5.212 (↑)	27.746 (↓)

34.722	0.35267	0.16546	0.37007	0.11783	4.934 (↑)	28.786 (↓)
41.666	0.35234	0.16516	0.36921	0.11666	4.788 (↑)	29.365 (↓)

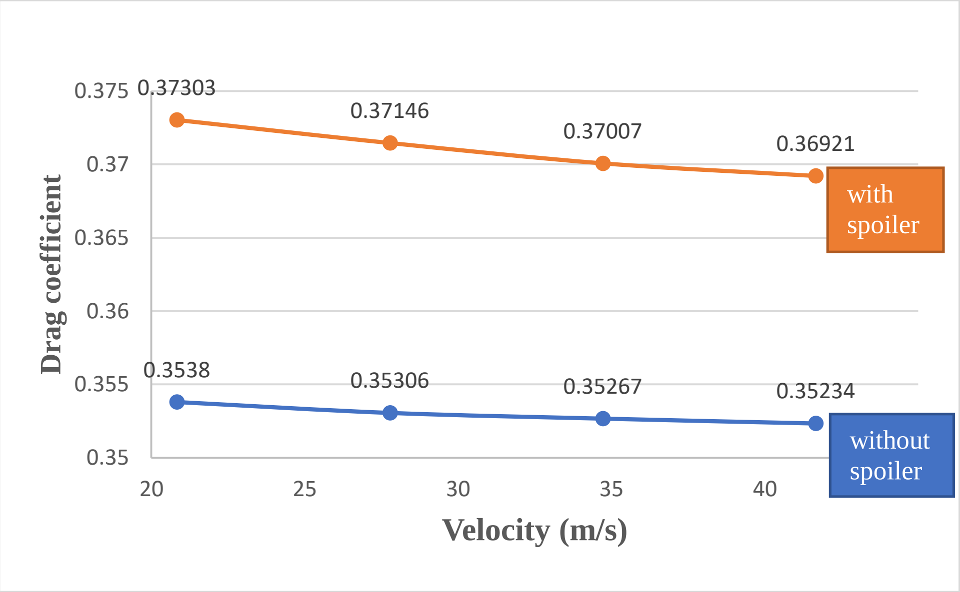
Table 2. C_d and C_l for both the car models

4. CONCLUSION

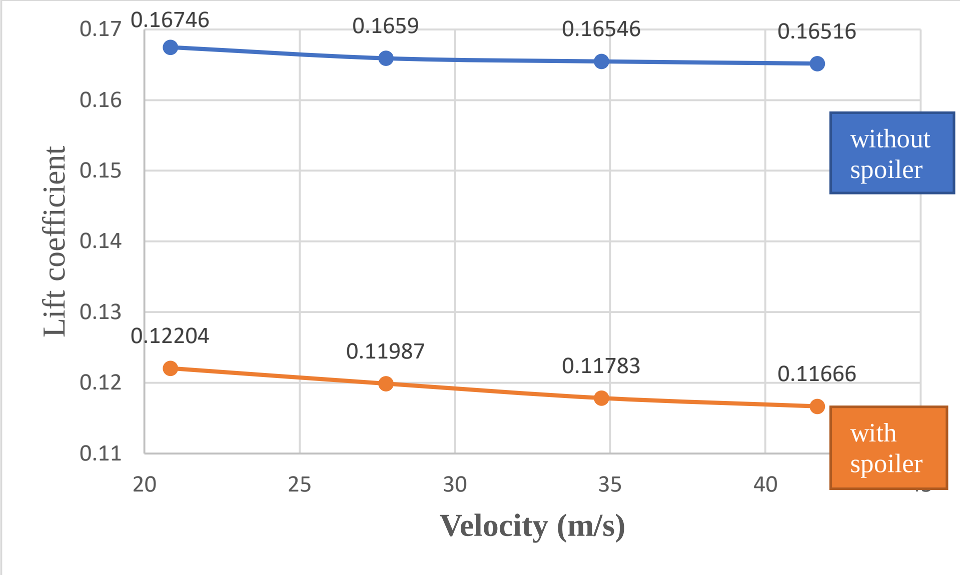
- It can clearly be seen from graph 1 that, the drag coefficient of the car is increasing by adding a spoiler. There is an average increase of 5.092% in the drag coefficient after adding thee spoilers in it.
- The results are converse in the case of lift coefficient. There is an average decrease of lift coefficient by 28.255%, after the addition of spoiler in the original model. This is depicted in the graph 2.
- One trend is similar in both the cases i.e. both C_d and L_d tend to decrease in value regardless of the car geometry. This decrease in values can be clearly observed in graphs 1 and 2.

4.1 Effect on fuel consumption:

Aerodynamic drag is proportional to the square of velocity, and hence the power needed to overcome drag is proportional to the cube of velocity. This means that there is a very strong relationship between the speed that a vehicle is travelling and the proportion of the fuel used to overcome drag. For passenger cars this means that aerodynamics is responsible for a much higher proportion of the fuel used in the highway cycle than the city cycle: 50% for highway versus 20% for city. This means that if you make a 10% reduction in aerodynamic drag your highway fuel economy will improve by approximately 5%, and your city fuel economy by approximately 2%.
Hence, the fuel consumption in our case would increase by 2.5% approximately (on highways) because the drag has increased by approximately 5%. But since the lift coefficient is decreasing approximately by 28%, the fuel consumption will also reduce by 14% approximately (on highways). Therefore, overall 11.5% fuel consumption reduction is observed and hence addition of spoiler has proved to be a fruitful experiment according to our test conditions.



Graph 1. Drag coefficient



Graph 2 Lift coefficient

5. FUTURE SCOPE

In our analysis we had taken the spoilers which were attached at a very negligible angle w.r.t. the car rear portion. We got an increase in drag coefficient while an increase in lift coefficient. Further, the spoiler can be played with and has a room for experimentation. May be a change in angle up to few degrees can result in decrease of both the coefficients. Overall it can be concluded that this area has a wide scope of research in future.

6. REFERENCES

[1] 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.
[2] 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.
[3] 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.
[4] 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.
[5] 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.
[6] 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.