Analysis Of The Spoiler

Parashar V. Shinde, Arshad Shaikh, Saad Mirza, Haidar Khan, Prof. Irshad Shaikh

Abstract—In this study we have done the aerodynamic study of a sedan vehicle. The simulation of the external aerodynamics of the vehicles is the most challenging and very much important automotive computational fluid dynamics application. With the speedy improvement in digital computers, computational fluid dynamics is used as a practical tool in present day practical research. In this study two different types of simulations were done. First simulation was done on a simple sedan car to find out the coefficient of drag and coefficient of lift then, the second simulation was done on the same simple sedan car model by installing the rear spoiler to find out the coefficient of drag and coefficient of lift. For all this simulation the base model was made in SolidWorks 2015 by considering the dimensions of the Chevrolet Cruze. This study result shows that the coefficient of drag and coefficient of lift of the vehicle model with spoiler has a mild reduction. This leads to less fuel consumption on the road, helps to maintain traction and increase braking stability.

Keywords—Computational Fluid Dynamics, Coefficient of drag, Coefficient of lift, 2D, Roof top

I. INTRODUCTION

After many years of the development in automobile aerodynamics the ideal water drop body shape, smooth under body, very less sharp corner, smooth upper surface finish and various methods have been applied in designing present day vehicle bodies. The flow of air over a vehicle determines the drag forces, which in turn affects the performance and also the efficiency of the car. Consider the car’s rear spoiler as an example. A spoiler is an automotive device whose intended design function is to ‘spoil’ unfavorable air movement across a body of a vehicle in motion, usually described as turbulence or drag. Spoiler is also sometimes called as wings. In a car a rear spoiler is most commonly installed on the trunk lid. This spoiler diffuses the flow of air passing a vehicle due to which turbulence is minimized at the rear of the vehicle or a car also it increases a downward pressure to the rear of the vehicle and reduces the lift. Some people must be thinking that the rear spoiler is only for decoration, but it do have the measurable effects on aerodynamic drag reduction and stability of the vehicle. This paper is focusing on the reduction of the aerodynamic drag and lift on the considered model. Earlier spoilers were mostly used for sports cars but nowadays with demand for increasing fuel efficiency it is also used on passenger cars. Hence, a passenger car model of Chevrolet Cruze was used for the simulation.

When a car body moves through the air it experiences an aerodynamic forces and moments from the air. Drag is the force acting on the car in direction opposite to the moving direction of the car. The force perpendicular to the drag and normal to the ground is called as lift. The drag force increases then the requirement of the horsepower also increases.

The coefficient of drag \(C_D\) and coefficient of lift \(C_L\) are shown below:

\[
C_D = \frac{\text{drag force}}{(\rho V^2 A)/2} \quad (1)
\]

\[
C_L = \frac{\text{lift force}}{(\rho V^2 A)/2} \quad (2)
\]

Where, \(\rho\) is density of air; \(V\) is the velocity of car; \(A\) is the frontal projected area of the car.
II. DESIGNING OF 2D CAD MODEL

At the beginning the car model dimensions of Chevrolet Cruze was selected for the preparation of the 2D CAD model as shown in Fig.1. The Chevrolet Cruze is the sedan car. We also selected the spoiler model dimensions for the preparation of the 2D CAD model of the spoiler as shown in the Fig.2.

**Figure 1. Dimensions of Chevrolet Cruze**

**Figure 2. Dimensions of spoiler**

Considering the dimensions of Chevrolet Cruze the simplified 2D CAD model of the vehicle was prepared which was without spoiler and then the second 2D CAD model of the vehicle having a rear spoiler on the trunk lid was prepared as shown in the Fig.3 and Fig.4 respectively. The 2D CAD model of the spoiler on the trunk lid of the vehicle was also prepared by considering the dimensions of the spoiler as shown in Fig.5. For preparing all the models SolidWorks 2015 was used.

**Figure 3. 2D simplified CAD model without spoiler**
III. CFD ANALYSIS OF THE MODELS

Computational Fluid Dynamics (CFD) is one of the most popular engineering analysis method. The CFD analysis was carried out by using Ansys 2015 in which fluent analysis system were used. As the models prepared in SolidWorks 2015 was 2D, so the analysis carried out will be also in 2D form. In the geometry the enclosure of specified dimensions was created surrounding the car 2D CAD model without spoiler. The same enclosure of the specified dimensions was created surrounding the car 2D CAD model with spoiler.

3.1. Meshing
On both the car body surface without spoiler or with spoiler the fine triangular shape surface mesh of 1 mm was done as shown in Fig.6 and Fig.7 respectively.
3.2. Boundary conditions
After meshing was done on both the car model without spoiler and with spoiler there were total five boundary conditions were applied.

- First boundary condition was inlet which was nothing but the velocity inlet and it was given 70 m/sec.
- Second boundary condition was outlet which was the pressure outlet.
- Third boundary condition was the boundary at top which was a fixed wall.
- Fourth boundary condition was the road on which the vehicle was stationary.
- Fifth boundary condition was the car body on which we have to find the coefficient of drag and coefficient of lift.

Figure 6. Fine meshing on surface of a car body without spoiler

Figure 7. Fine meshing on surface of a car body with spoiler
All these above boundary conditions were given to the model of a car without spoiler, as well as to the model of car with spoiler. The boundary conditions of the model of a car without spoiler are shown in Fig.8 and the boundary conditions of the model of a car with spoiler are shown in Fig.9 below.

**Figure 8. Boundary conditions given to model of car without spoiler**

**Figure 9. Boundary conditions given to model of car with spoiler**

### 3.3 Setup for analysis

For doing the analysis the time is kept steady. The viscous – standard k-omega models were preferred for the analysis. The fluid used was air. The 70 m/sec value was given to the inlet boundary condition, which means we are keeping the car as stationary and giving the velocity of 70 m/sec to the flow of air across the car body. The solution method pressure velocity coupling scheme used was coupled as it takes more time but gives more accurate results. At last we have to run the iterations to get the result. The iterations should be runned until the solution is converged to give the values of $C_D$ and $C_L$. 
3.4 Results
The contours of static pressure for both the 2D CAD models of car without spoiler and with spoiler are shown in Fig.10 and Fig.11 respectively. From figures we can clearly see that the maximum pressure is in the front side of the hood of the car as it is represented in red color. So, the maximum pressure acting on the car without spoiler is $3.24 \times 10^3$ which is equal to 3237.776 Pa and the maximum pressure acting on the car with spoiler is $3.24 \times 10^3$ which is equal to 3236.886 Pa.

![Figure 10. Contours of static pressure for model of car without spoiler](image1)

![Figure 11. Contours of static pressure for model of car with spoiler](image2)

The contours of velocity in X-direction were shown on both the 2D CAD models of car without spoiler and with spoiler in Fig.12 and Fig.13 respectively. From figures we can clearly see that the maximum velocity is present over the roof top of the car as it is indicated in red color. So, the maximum velocity present on the roof top of the car without spoiler is $1.21 \times 10^2$ which is equal to 120.7896 m/sec and the maximum velocity present on the roof top of the car with spoiler is $1.25 \times 10^2$ which is equal to 124.5325 m/sec.
3.4.1 Comparison of the graphs
The graphs of coefficient of drag ($C_D$) and coefficient of lift ($C_L$) was plotted for both the car models i.e. for car model without spoiler and car model with spoiler. The graph of $C_D$ and $C_L$ for car model without spoiler is shown below in Fig.14 and Fig.15 respectively. The graph of $C_D$ and $C_L$ for car model with spoiler is shown below in Fig.16 and Fig.17 respectively.
Figure 14. $C_D$ for car model without spoiler

Figure 15. $C_L$ for car model without spoiler
By comparing the graph shown in Fig.14 and Fig.16, we can observe that the value of drag shown in Fig.14 without spoiler is 0.555 whereas the value of drag shown in Fig.16 with spoiler is 0.486. So we can say that after installing the spoiler the drag has been decreased.

By comparing the graph shown in Fig.15 and Fig.17, we can observe that the value of lift shown in Fig.15 without spoiler is 0.503 whereas the value of lift shown in Fig.17 with spoiler is 0.108. So we can say that after installing the spoiler the lift has also been decreased.

The comparison of Drag and Lift is shown in Table.1 below.
Table 1. Comparison of Drag and Lift

| Sr. No. | Model       | $C_D$ | $C_L$ |
|--------|-------------|-------|-------|
| 1      | Without spoiler | 0.555 | 0.503 |
| 2      | With spoiler  | 0.486 | 0.108 |

IV. CONCLUSION

The aerodynamic lift, drag and flow characteristics of a high speed sedan car were numerically investigated. It is observed that the drag force and the lift force at the back side of the car were reduced by installing the rear spoiler or wing. In short, the analysis reveals that the rear spoiler have considerable effect on lift and drag of the vehicle which has the following advantages:

1. Increases vehicle stability at high speed.
2. Reduces fuel consumption.

REFERENCES

[1] R. B. Sharma, Ram Bansal, Aerodynamic Drag Reduction of a Passenger Car Using Spoiler with VGs, International Journal of Engineering Research and Application (IJERA) ISSN 2248-9622, March 2014.
[2] J. Godwin John, A. Muthuvel, Numerical Analysis for Drag Reduction in Commercial Buses, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN 2278-1684, p-ISSN 2320-334X, PP 01-05, 2014.
[3] Aniruddha Patil, Siddharth Kshirsagar, Study of Front Wing of Formula One Car using Computational Fluid Dynamics, International Journal of Mechanical Engineering and Robotics Research (Int. J. Mech. Eng. & Rob. Res.) ISSN 2278-0149, Vol. 3, No. 4, October 2014.
[4] Akshay Parab, Ammar Sakarwala, Aerodynamic Analysis of a Car model using Fluent-Ansys 14.5, International Journal on Recent Technologies in Mechanical and Electrical Engineering (IJRMEE), ISSN 2349-7947, Vol. 1, Issue 4, November 2014.
[5] A. Sunanda, M. Siva Nayak, Analysis of NACA 2412 for Automobile Rear Spoiler using Composite Material, International Journal of Emerging Technology and Advanced Engineering (IJETAE), ISSN 2250-2459, Vol. 3, Issue 1, January 2013.
[6] Shyam P. Kodali, Srinivas Bezavada, Numerical Simulation of Air Flow Over a Passenger Car and the Influence of Rear Spoiler using CFD, International Journal of Advanced Transport Phenomena (IJATP), PP 6-13, Vol. 01, No. 01, Jan-Dec 2012.
[7] Xu-xia Hu, Eric T.T. Wong, A Numerical Study on Rear-Spoiler of Passenger Vehicle, International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering, Vol. 5, No. 9, 2011.