Aerodynamics analysis of the car using Solidworks flow simulation with rear spoiler using CFD

L. Prabhu¹, Sangeetha Krishnamoorthi¹, P Gokul², Nandhu Sushan², P H Hisham Harshed² & Aviss Jose²

¹Associate Professor, Department of Mechanical Engineering, Aarupadai Veedu Institute of Technology, Vinayaka Mission Research Foundation, Deemed to be University, Tamil Nadu, India.
²UG Scholar, Department of Mechanical Engineering, Aarupadai Veedu Institute of Technology, Vinayaka Mission Research Foundation, Deemed to be University, Tamil Nadu, India.
Email: prablogu@gmail.com

Abstract. The Vehicle stability is the major concern in the high-speed car being produced now a day. The fuel consumption is mainly affected by different forces which include the force due to drag and lift, force due to self-weight, side forces and also the thrust acting on the vehicle. About 60% of the drag is which is caused due to the relative motion between the air and the vehicle is created at the rear end. The fuel consumption can be increased by reducing the drag force at the rear end. So, this paper aims to reduce the drag force which helps in improving the fuel consumption which leads to environment protection. We used a Sedan car with various types of spoilers for reducing the aerodynamic drag force. The design of the car was done on SOLIDWORKS and also it was used for the analysis in SOLIDWORKS FLOW SIMULATION. The analysis was done to find out the drag and lift forces at various velocities and by using different spoilers. This study proposed a numerical model which was effective when done on the computational fluid dynamics (CFD) so as to obtain the different flow structure around the car with spoiler at the rear end.

Keywords: Rear spoiler, Drag and lift force, Solid works flow simulation.

1. Introduction:

The new cars which are being manufacture in the current trends need to be sportier for the customers to procure it. The current aerodynamic properties are not sufficient to cope up with the high power and speed and it is not able to accept the down force received. The aerodynamic properties play a major role not only in the performance of the vehicle but also in the safety and comfortness for the passengers. In order to maintain the stability and reduce the drag force developed few auxiliary parts like rear spoilers, bumpers at the front and rear, are being added to enhance a proper air flow. During high speed the Aerodynamic properties reduces the vehicles performance because of the lift and drag forces developed. The literature reviews have described that the wind tunnel testing has been a time consuming process and with the development of computer technology the CFD can be a good alternative for reducing the time required for testing. The analysis on the flow of air was done to find the lift and drag force on the vehicle by using Ansys 14.5 and later it was done with the diffuser provided at different angles. Then based on the Ci and Cd values the best model was selected [1]. The air movement effects towards the solid objects were found out by using the Wind tunnel testing.
machine. The main part of the wind tunnel was the testing chamber and the drag and lift forces were calculated based on this. The beam balances which is mainly connected with the help of beams or cables helps to measure the aerodynamic forces [2]. The lap time could be achieved in the race car by reducing the drag force and sometimes an additional drag force is also developed because of the air flow through the passage provided for the radiator. By having an angle of tilt around 72.5 degree in the cooling channel there was a reduction in the drag force of about 12.7% [3]. The streamline shape or the continuing curvature is the main reason for pressure to be low on the down over nose and the trunk. High pressure contours were found at the radiator zone in the front and the pressure contours which were obtained by using CFD were close to the values which were obtained by conducting the experiment [4]. The commercially available CFD code Fluent with the k€ turbulence model was used for calculating the simulations of Aerodynamics and the LES turbulence model was mainly used for finding out the aero acoustics. The 2 step process using low Mach number was used for finding the aerodynamics and aero acoustics. This method reduced the time required for calculating the analysis of noise and also showed improvement in the accuracy of prediction [5]. From CFD and wind tunnel calculations the coefficient of drag was found to be 0.42 to 0.48 and the difference was about 12.5%. This is due to the reason that there is an inviscid flow modelling which does not consider the friction of skin and so there exist a boundary layer on the panel of the body. The drag coefficient occurs due to the tyres at an average of about 25.7%. The down force is produced in the car without considering the tyres and a lift force is obtained when the tyre are considered. The aerodynamics drag requires power of about 20% when the car is running at a speed of 40 km/hr and around 6% of the total amount of fuel used by the car [6]. Each component on the vehicle is equally responsible for bringing the aerodynamic force to a steady state and more time was required for the lift force, pitch and roll moments. The under body flow have a transient reaction and is the main reason for the longer relaxation time and is totally represented by the modification and separation surrounding the windward wheels [7]. The mathematical equations in the partial differentiation type are generally used to analyse the physical characteristics of the fluid motion [8].

2. Objective

When the speed limit is about 110 km/h in the highway there is a possibility of the car to lift over. The high pressure air which occurs in the front side of the windshield causes the pressure to drop and the lift may occur. This low pressure is transferred to the car’s roof when the air passes over it. After passing through it the air manages to pass the rear window where will be a notch produced due to the window dropping down on to the trunk and will leave a vacuum or low pressure which cannot fill the space properly. Due to this a low pressure a lift is created and the flow will be detached and then will act on the trunk’s surface area. The main objective of this paper is to analyse what happens when a rear spoiler is attached on the trunk to create a high pressure when there is a lift on the trunk. In Sedan type cars spoilers can be used easily. These spoilers act like a battery to air flow and increase the amount of air pressure in front the spoiler. The Sedan type cars which are light in the rear face the lift easily and so the spoilers are used and the various effects of shear stress analysis, pressure analysis, lift and drag force analysis are found out.

3. Methodology

3.1Drag Force

The force that acts in an opposite direction to the relative motion between any object that moves with respect to the fluid is called drag force. This exists between 2 fluid layers and even in solid surfaces. The drag force and lift force of the vehicle is proportional to the drag and lift coefficient respectively [9]. The formula for finding the drag force which is experienced by the object due to the movement of fluid is given by $F_D = \frac{1}{2} \rho V^2 C_D A$, where $C_D$ is the dimensionless coefficient called as drag coefficient and mainly depends on the Reynolds number. The drag force acting on the car is shown in Fig 1.
3.2 Lift Force

A force is exerted on the body when the fluid passes over it. The force that acts in a perpendicular direction of the fluid flow is called as lift force whereas the drag force acts in a direction parallel to it. This force is called as aerodynamic force when the fluid used is air and hydrodynamic force if it uses water. The lift force developed for the given specific flow conditions is given by the equation \( \text{L} = \frac{1}{2} \rho V^2 AC_L \), where \( C_L \) is the lift coefficient at the desired angle of attack. The direction in which the lift force acts is shown in the lift force theory given in Fig 2.

4. Vehicle generic models and dimensions

Both the front and the rear section of the car are subjected to coefficient of lift. The overall surface of the new sedan car is first scanned and then exported in the SOLIDWORKS software for doing the smoothness and repair the surface if any required. Then the CFD analysis is done using the Star-CFD software. At the center place of the car the pressure distribution data and velocity vector is plotted by the post processor. Since the center portion of the car will be more similar to the outline of the original shape of the car it is selected. When the pressure decreases along the streamline we can identify the pressure distribution as a favorable one for analyzing [10]. Then the pressure distribution and velocity vector pattern is used to find out the effect of drag force and lift force acting. The side view and top view of the generic vehicle model is shown in Fig 3 and Fig 4 respectively. The Step by step modeling of car in surface modeling using SOLIDWORKS is done and the step of inserting the images is shown in Fig 5.

Fig 1: Drag force on a car

Fig 2: Lift force theory

Fig 3: Side view of the car

Fig 4: Top view of the car

Fig 5: Step by step modeling of car in surface modeling using SOLIDWORKS
5. Spoiler Generic Models and Dimensions

For analyzing numerically the 2 different spoilers were used. The first spoiler also called wing type spoiler was mounted at a height of about 28cm from the vehicle’s rear end surface and the second spoiler was placed on the edge of the vehicle’s rear side without any gap between the surface of the vehicle and the spoiler.

Fig 3: Side view of the generic model vehicle  
Fig 4: Top view of the generic model vehicle  
Fig 5: Inserting Images  
Fig 6: Generic model and dimensions of first spoiler  
Fig 7: Generic model and dimensions of second spoiler
Among the 2 type of spoilers the wing type spoiler is efficient as it reduces the drag coefficient than increasing the lift coefficient [11]. The generic model and dimensions of the first and second spoiler is shown in Fig 6 and Fig 7 respectively. The assembled view of the vehicle with both the spoilers placed in position is shown in Fig 8 and Fig 9 respectively.

**Fig 8:** Assembly 3D CAD model of vehicle and first spoiler  
**Fig 9:** Assembly 3D CAD model of vehicle and second spoiler

### 6. Virtual wind tunnel and vehicle orientation

To represent the wind tunnel as it applies in real cases a virtual air box was created around the 3D model. Since the spoilers are going to be placed in the rear and we are going to analyze the different forces at the rear end large amount of space is left at the rear end. This is also done to find out the flow behavior at the back side of the vehicle. The virtual wind tunnel created in the 3D model is illustrated in Fig 10.

**Fig 10:** Virtual wind tunnel and vehicle orientation

As there was a lot of complexity involved in the simulation and also with the limited number of resources available the full domain was divided in to half by using an YZ symmetry plane in which the simulation can be done in one half as the other side is also symmetric. To make the slip wall with no shear force a symmetric boundary was defined in the solver’s YZ plane due to which the results obtained from simulation will be considered for full model. To make the SOLIDWORKS flow simulation identify and apply the appropriate boundary conditions a name has been given to all the 5 surfaces in the virtual wing tunnel created. The final mesh created is shown in Fig 11. This same steps was repeated for creating a high resolution mesh for all the 3
cases namely i) the vehicle itself ii) the vehicle with first spoiler iii) the vehicle with second spoiler. The different parameters for the meshing are given in Table 1.

![Image: The Final mesh](image)

**Table 1: Mesh Sizing Parameters**

| FACE                          | BOUNDARY TYPE   | ZONE NAME   |
|-------------------------------|-----------------|-------------|
| In front of the car           | Velocity Inlet  | Inlet       |
| Side opposite the Car         | Symmetry        | Side        |
| Above the Car                 | Symmetry        | Top         |
| Below the Car                 | Wall            | Ground      |
| Two faces created at the car center plane | Symmetry | Symmetry Plane |
| Behind the car                | Pressure Outlet | Outlet      |
| Faces of the car              | Wall            | Car         |

**7. CFD Analysis**

The process used for simulating the performance of systems, equipment's involved in the flow of gases as well as liquids, the amount of heat and mass transferred and the chemical reactions taking place among them is called Computational fluid dynamics. To reduce the drop in pressure and to find out the aerodynamic lift and drag pressure the process of CFD is used. CFD is also used to find out the thrust produced in the rotor and to calculate the air flow and also to ensure proper cooling occurs in the system. SOLIDWORKS flow simulation is one of the best processes in the fluid simulation and is used when the product is to be manufactured with safety and good performance.
7.1 Velocity Diagram with respect to various Speeds

The velocity diagram without spoiler at different speeds of 70, 90, 110, 130, 150 kmph is shown in the Fig 12(a), Fig 12(b), Fig 12(c), Fig 12(d) and Fig 12(e) respectively.

![Velocity Diagrams (a) to (e)]

**Fig 12:** Velocity diagram at (a) 70kmph (b) 90kmph (c) 110kmph (d) 130kmph (e) 150kmph

7.2 Pressure Analysis with respect to various Speeds

The Pressure analysis at various speeds of 70, 90, 110, 130, 150 kmph is shown in Fig 13 (a), Fig 13(b), Fig 13(c), Fig 13(d) and Fig 13(e) respectively.
7.3 Shear Stress analysis with respect to various Speeds

The shear stress analysis at different speeds of 70, 90, 110, 130, 150 kmph is shown in Fig 14(a), Fig 14(b), Fig 14(c), Fig 14(d) and Fig 14(e) respectively.
7.4 Drag & Lift force analysis with respect to various Speeds

The drag and lift force analysis at different speeds of 50, 70, 90, 110 and 130 kmph is shown in Fig 15(a), Fig 15(b), Fig 15(c), Fig 15(d) and Fig 15(e) respectively.
8. Results and Discussion

This paper discusses about the aerodynamic drag and lift forces by analyzing without a spoiler, analyzing with a spoiler 1 and spoiler 2 at the rear end. The analysis started first without using a spoiler and in that the pressure analysis, shear stress analysis and the lift and drag force analysis was done. Later the same tests were done using the spoiler 1 and spoiler 2 and the data has been tabulated below. The drag and lift force without using a rear spoiler is shown in Table 2.

Table 2: Car without rear spoiler

| Velocity (Km/h) | Drag Force (N) | Lift Force (N) |
|-----------------|----------------|----------------|
| 70              | 251.408        | 112.47         |
| 90              | 415.813        | 187.86         |
| 110             | 631.27         | 282.51         |
| 130             | 861.27         | 383.67         |
| 150             | 1150.36        | 526.42         |

Similarly by using the spoiler 1 and spoiler 2 the values are presented in Table 3 and Table 4 below respectively.

Table 3: Car with rear spoiler 1

| Velocity (Km/h) | Drag Force (N) | Lift Force (N) |
|-----------------|----------------|----------------|
| 70              | 244.82         | 71.17          |
| 90              | 403.702        | 114.86         |
| 110             | 612.23         | 170.06         |
| 130             | 844.29         | 237.768        |
| 150             | 1123.34        | 336.48         |
Table 4: Car with rear spoiler 2

| S. No | Velocity (Km/h) | Drag Force (N) | Lift Force (N) |
|-------|-----------------|----------------|---------------|
| 1     | 70              | 225.20         | 67.22         |
| 2     | 90              | 369.26         | 113.45        |
| 3     | 110             | 554.88         | 169.46        |
| 4     | 130             | 778.02         | 234.32        |
| 5     | 150             | 1037.03        | 312.768       |

9. Conclusion

The CFD analysis was done on the car selected for study and the results obtained were close to the officially given values. As the simulation was over the next step would be to do modifications in the design due to which there may be change in the performance of the vehicle. As per the law of fluid dynamics the velocity of the fluid will increase and the pressure will decrease when it pass through the constriction. The CFD analysis started with the results obtained from SOLIDWORKS simulation and gave the different values of drag force and lift force at different velocities. The CFD simulation values presented in the results show us that the car when using a second spoiler showed better results even as the velocity goes on increasing. From the CFD analysis it is clear that the aerodynamics properties can be further improved by having a change in the exterior model of the car body which helps in reducing the coefficient of drag also. So as a conclusion by introducing a spoiler at the rear without any gap between the surfaces led to better driving conditions with good stability.

10. References

[1] Akshay Parab et al., 2014, Aerodynamic Analysis of a Car Model using Fluent- Ansys 14.5, International Journal on Recent Technologies in Mechanical and Electrical Engineering, Vol-1, Issue-4, 007-013.
[2] Ashfaque Ansari et al., 2014, Drag Force Analysis of Car by Using Low Speed Wind Tunnel, International Journal of Engineering Research and Reviews, Vol. 2, Issue 4,144-149.
[3] Chung Sun Lee et al., 2014, Calculation and Optimization of the Aerodynamic Drag of An Open-Wheel Race Car, Eureka 13, 29-30
[4] Bhagirathsinh H Zala et al., 2015, Comparative Assessment Of Drag Force Of Sedan Car Model By Computational Fluid Dynamics And Experimental Method, International Journal of Advanced Engineering Technology, Vol. VI/Issue 1, 06-10
[5] Chien-Hsiung Tsai et al., 2009, Computational aero-acoustic analysis of a passenger car with a rear spoiler, Applied Mathematical Modelling, 33, doi:10.1016/j.apm.2008.12.004,3661–3673
[6] Rizal E. M. Nasir et al., 2012, Aerodynamics of ARTeC’s PEC 2011 EMo-C Car, Procedia Engineering, 41, doi: 10.1016/j.proeng.2012.07.382, 1775 – 1780
[7] Makoto Tsubokura et al., 2010, Large eddy simulation on the unsteady aerodynamic response of a road vehicle in transient crosswinds, International Journal of Heat and Fluid Flow, 31, doi:10.1016/j.ijheatfluidflow.2010.05.008,1075–1086
[8] V. Naveen Kumar et al., 2015, Investigation of Drag and Lift Forces over the Profile of Car with Rearspoiler using CFD, International Journal of Advances in Scientific Research, 1(08): 331-339.
[9] Mohankumar M et al., 2015, Investigating the Effect of Rear Spoiler and Rear Diffuser on Aerodynamic Forces using CFD, International Journal of Engineering Research & Technology, Vol. 3, Issue 26, 1-6.
[10] Shashank Arya et al., 2016, A Review on “Reduction of Drag Force using ADD-ON Devices”, International Research Journal of Engineering and Technology, Vol. 03 Issue 11, 985-994.
[11] Shivaji S. Shinde et al., Enhancement of Aerodynamic Drag Reduction of Passenger Vehicle using CFD analysis-Review, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 6, Special Issue 1, 437-448.