INVESTIGATION ON AUTOMOTIVE AERODYNAMIC DRAG REDUCTION OF A HATCHBACK CAR BY REAR SPOILER USING CFD ANALYSIS

M. SATHISH KUMAR¹, ARUN NEMA² & S. JAICHANDAR³

^{1,3}Department of Automobile Engineering, Vel Tech Rangarajan Dr. Sagunthala R&D
Institute of Science and Technology, Avadi, Chennai, Tamil Nadu, India

²Department of Aeronautical Engineering, Vel Tech Rangarajan Dr. Sagunthala R&D
Institute of Science and Technology, Avadi, Chennai, Tamil Nadu, India.

ABSTRACT

This research paper concerns to optimize the various aerodynamic characteristics of the vehicle to reduce the drag resistance and to increase the fuel efficiency and vehicle performance. Nowadays, fuel consumption of the vehicle is the major problem to deal. So, to reduce the fuel consumption and to improve the fuel economy, aerodynamic drag resistance will be considered and minimized. In this project, the drag resistance of the vehicle will be reduced by using the rear spoiler. The spoiler is designed for achieving the best result in reducing the drag in a hatchback car model. The scaled model of the car is created by using the software CATIA V5R20. The spoiler also designed using this software. The Computational Fluid Dynamics (CFD) simulation and analysis is done on the scaled modeled car with the spoiler by using ICEM CFX. The correlation between the without and with spoiler arrangements on the scaled model car is also investigated.

KEYWORDS: *Automobile Aerodynamic Characteristics, Rear Spoiler, Drag Reduction, Fuel Efficiency Improvement & Vehicle Performance Improvement.*

Received: Mar 02, 2018; **Accepted:** Mar 22, 2018; **Published:** Apr 10, 2018; **Paper Id.:** IJMPERDAPR2018156

INTRODUCTION

In the present world, improving the fuel efficiency and reducing the fuel consumption of the vehicle is the challenging criteria. This can be achieved by considering the various properties of the vehicle while running. The improved aerodynamic property of the vehicle by using add-on devices makes a lot of improvement in the fuel efficiency. By reducing the drag of the vehicle the fuel efficiency will be improved. The better aerodynamics of the vehicle by add-on devices results in reducing drag, wind noise, minimizing noise emission. This will avoid the undesired lift forces and other causes of aerodynamic instability at high speeds. Another important thing is it will produce the downforce to improve the traction and thus cornering abilities.

SPOILERS

When the vehicle is in motion, the device which is used to spoil the turbulence effect of air is known as the spoiler. The front spoiler and rear spoiler is otherwise called as front wing and rear wing. These devices will be attached to the front part and rear part of the vehicle.

In the olden days, researchers are concentrated more on the development of the engine technology for the purpose of racing. After that, they concentrate on advancements in a tire. Later the importance of the race car aerodynamics is evolved. At that time the aerodynamics technology is not much expensive. Now the technology has been developed and they are testing the aerodynamic effects of the vehicle in the expensive wind tunnels. In the year 1972, the front and rear spoilers were efficiently used to gain the performance of the vehicle.

Design of the Car and Spoiler

Design and Dimensions of the Hatchback Car

The solid model of the car has been designed in the ratio of 1:10 with respect to the given dimensions of the specifications using the software CATIA V5R20.

Table 1

SPECIFICATIONS	DIMENSIONS (mm)
Overall Length (mm)	3595
Overall Width (mm)	1650
Overall Height (mm)	1900
Wheel Base (mm)	2400
Ground Clearance (mm)	165
Front Track (mm)	1295
Rear Track (mm)	1290

Figure 1: Solid Model of the Hatchback Car

Design and Dimensions of the Spoiler

A newly designed spoiler is used in this research. It is going to be analyzed with hatchback model car by fixing at the rear side.

The design and dimensions of the rear spoiler is as follows:

Figure 2: Dimensions of Spoiler

CFD ANALYSIS OF A MODELED CAR WITHOUT SPOILER

Geometry Creation and Meshing

Figure 3: Creation and Meshing

In general, Mesh can be classified as the surface mesh or volume mesh. Based on the shape they can be classified as tetrahedral, hexahedra, quad, tetra and prism. The tool used in our case to achieve the required mesh is ICEM CFD of Ansys 13. Initially, geometry clearance is said to be done to remove unattached curves before the mesh is generated.

- Geometry clearance tolerance of 0.4 is used.
- Tetrahedral volume mesh is used.

Boundary Condition

Table 2

Analysis Type	Steady State
Fluid	Air at 25°C
Heat transfer	Isothermal
Buoyancy model	Non buoyant
Domain motion	Stationary
Turbulence model	K epsilon
Buoyancy model	Non buoyant
Reference pressure	1 atm.
Inlet flow speed	20 m/s
Outlet relative pressure	0 pa
Side walls	Free slip wall boundaries
Upper and lower walls	Free slip wall boundaries
Car body	No slip wall boundary
Car wall roughness option	Smooth wall

Figure 4: Model Car with Boundary Conditions

RESULTS AND DISCUSSIONS

Figure 5: Plots of Pressure and Velocity

From the contour plots, it can be seen that near the surface of the car there is almost zero velocity this can be attributed to the boundary layer phenomenon. Similarly, over the roof of the car, it is seen that flow of air is said to be accelerated and this created an area of low pressure on the car. Near the bumper of the car, there is seen to be an area of very high pressure this region is said to be called as the stagnation point and the velocity of the fluid reaches zero in this area. There is a region of low fluid velocity behind the boot of the car this is the wake of the vehicle. This region of low velocity extends for a certain distance after which the flow regains its original stream velocity in a progressive manner.

The analysis was repeated for five different fluid speeds and the drag, lift forces and their coefficients are said to be observed.

Variable Tabulation for a Car without Spoiler

Table 3: Variable Tabulation for a Car without Spoiler

Velocity(M/S)	Drag Force(N)	Lift Force(N)	Drag Coefficient	Lift Coefficient
4	0.9516	0.0442	0.61728	0.02975
8	1.5910	0.1846	0.60782	0.03101
12	2.5951	0.4649	0.60602	0.04811
16	4.5885	1.3337	0.60411	0.05600
20	6.9451	1.4377	0.60301	0.03864

Graphical Plots for a Car without Spoiler

Figure 6: Graphical Representations for a Car without Spoiler

From the plots, it can be seen that the drag and lift forces keep increasing with the inlet velocity and both the lift coefficient initially show a slight increase with velocity but later they decrease consistently with the further increase in inlet velocity, while the drag coefficient decreases consistently.

CFD ANALYSIS OF A MODELED CAR WITH SPOILERS

Drag Reduction Mechanism Induced by the Rear Spoiler Device

The rear part of the car undergoes difference in pressure at the upper flow, side flow and underflow. The flow which is on the top surface is having higher pressure when compared to side flow and bottom flow. The flow will be in streamline motion. But the rear spoiler device blocks out the upper flow. Then the flow of air in the side surface and bottom surface will occupy the rear surface of the vehicle. Anyway, the pressure will be more on the top surface of the vehicle. Finally, the downforce is created due to the increased upper pressure, which increases the stability of the vehicle and results in the reduction of aerodynamic drag.

Modeled Car with Spoiler

Geometry Creation and Meshing

Figure 7: Meshing Of Car and Spoiler

Boundary Conditions

Figure 8: Boundary Conditions for a Car with Spoiler

Expressions Defined

Table 4

Variable	Definition	Expression
U_{inf}	Inlet velocity	20m/s
F_x	Drag force	force_y()@car
F_y	Lift force	force_z()@car
Denom1	Denominator 1	$0.5 * \text{massFlowAve}(\text{Density}) @ \text{inlet} * U_{inf}^2 * 0.0494$
Denom2	Denominator 2	$0.5 * \text{massFlowAve}(\text{Density}) @ \text{inlet} * U_{inf}^2 * 0.1574$
C_d	Coefficient of drag	$F_x / \text{Denom1}$
C_l	Coefficient of lift	$F_y / \text{Denom2}$

RESULTS AND DISCUSSIONS

Figure 9: Plots of Pressure and Velocity of a Car with Spoiler

Table 5: Variable Tabulation of Car with Spoiler

Velocity(m/s)	Drag force(N)	Lift force (N)	Drag coefficient	Lift coefficient
4	0.24791	-0.00322	0.35901	-0.00216
8	0.97523	0.00632	0.35273	0.00106
12	2.19968	0.25703	0.34926	0.02013

Table 5: Contd.,				
16	3.95165	0.74000	0.34757	0.03107
20	5.97969	0.16543	0.34566	0.00444

Graphical Representation of a Car with Spoiler

Drag Force Vs Velocity lift Force Vs Velocity

Drag Coefficient Vs Velocity lift Coefficient Vs Velocity

Figure 10: Graphical Representation of a Car with Spoiler

Generating the Spoiler Effect

The drag force and the lift force should be considered while designing the spoiler. Because to identify the effect of the spoiler, the drag force and lift force should be reduced. The two main components of the aerodynamic drag force is friction drag and pressure drag. Friction drag is based on the shear stresses on the vehicle body. Pressure drag is based on the pressure differences between front and rear part of the vehicle. If the lift force is positive, the value should be very small. To generate the good spoiler effect, the spoiler should be designed to create a negative lift force. The downforce is generated because of the high pressure on the top surface of the rear spoiler. And it has proven that the turbulence will be less behind the car.

Comparison Graphical Representation of without Spoiler and with Spoiler of a Car Model

Drag Force Vs Velocity lift Force Vs Velocity

Drag Coefficient Vs Velocity lift Coefficient Vs Velocity

Figure 11: Comparison Graphs of Without Spoiler and With Spoiler

Comparison of Rear Spoiler Drag Coefficient

Table 6: Comparison of Rear Spoiler Drag Coefficient

Velocity (m/s)	Without Spoiler	With Spoiler
	Drag Coefficient (C_D)	Drag Coefficient (C_D)
4	0.61728	0.35901
8	0.60782	0.35273
12	0.60602	0.34926
16	0.60411	0.34757
20	0.60301	0.34566

Drag Coefficient

The aerodynamic drag coefficient is a measure of the effectiveness of a streamline aerodynamic body shape in reducing the air resistance to the forward motion of a vehicle. A low drag coefficient implies that the streamline shape of the vehicle's body is such as to enable it to move easily through the surrounding viscous air with the minimum of resistance.

From the comparison graphs of without and with spoilers, it can be seen that the lift force, lift coefficient, drag force, a drag coefficient of a car without spoiler is increasing consistently. But with spoilers, it is reduced drastically. The shape of designed spoiler is giving effective results in drag reduction. Especially the drag coefficient is much reduced in a car with the spoiler. In this research analysis, the ultimate aim is to reduce the aerodynamic drag coefficient to improve the fuel efficiency by using the rear spoiler.

CONCLUSIONS

By designing the spoiler, to reduce the drag resistance of the vehicle will be quiet possibly achieved. The computational fluid dynamics analysis shown the different contour pressure and velocity, streamline pressure and velocity to understand the high pressure and low-pressure regions of the vehicle. In CFD analysis, near the bumper of the car, there will be a highpressure region called stagnation point and the velocity of the fluid will be zero in this area. The drag coefficient of the car, will be minimum after attaching the rear spoiler when comparing the car without the spoiler. The car with spoiler indicates the drag reduction will be less. But the comparison of the CFD result analysis, made the spoiler design is having good efficient to reduce the drag.

- When the vehicle is traveling at a high speed, the fuel efficiency is achieved by reducing the drag of the vehicle using the rear spoiler.
- The Drag co-efficient of the vehicle is around 0.34.
- The high-pressure zone is observed in the bonnet area of the car.

REFERENCES

1. "Aerodynamics of road vehicles" 'from Fluid mechanics to vehicle Engineering' – Wolf Heinrich Hucho, Germany, 1993.
2. "Fundamentals of Vehicle Dynamics" – Tomas Gillespie March 1992/ ISBN-10: 1560911999 / ISBN-13: 978-1560911999
3. "Advanced Vehicle Technology" – second edition, Heinz Heisler, Butterworth Heinemann, London, 2002, Pages 584 to 634
4. "An Experimental Investigation of flow over the rear end of a notchback automobile Configuration", Luther N. Jenkins NASA, Langley research center. SAE Technical Paper 2000-01-0489, 2000, doi:10.4271/2000-01-0489.
5. "Twenty years of projects on vehicle aerodynamics" - A. S. WHITE, Middlesex University, Bounds Green Road, London N11 2NQ, UK
6. Pratiksha Nallawar, Vinayak Narwade et al., Design and Analysis of Intake Manifold Using 3-D CFD Analysis, International journal of Mechanical and Production Engineering Research and Development (IJMPERD), Volume 5, Issue 5, September - October 2013, pp. 115-122
7. "Formula SAE Vehicle Aerodynamics" - Matthew R. James, Thesis report 2008.
8. "Research on Aerodynamic Drag Reduction by Vortex Generators" - Masaru KOIKE* Tsunehisa NAGAYOSHI* Naoki HAMAMOTO*Mitsubishi motors, Technical review 2004.
9. A comparative assessment of two experimental methods for aerodynamic performance evaluation of car. Manan Desai, S A Channiwala and H J Nagarsheth. Sardar Vallabhai National Institute of Technology, Surat. Issue 4, volume 3, oct 2008.
10. Hoerner, S. F., Fluid-dynamic Drag, Published by the author, 1958
11. Shibata, H., MMC's Vehicle Wind Tunnel, Automobile Research Review (JARI) Vol. 5, No. 9, 1983