

# VEHICLE AERODYNAMIC ANALYSIS – A Primitive Approach

P. Sivakumar<sup>1</sup>, N. Rajalakshmi<sup>2</sup>, M. Shalini<sup>3</sup>, N. Malathi<sup>4</sup>

<sup>1</sup>Assistant Professor, Department of Aeronautical Engineering,  
Sri Ramakrishna Engineering College, Tamil Nadu, India

<sup>2,3,4</sup>Department of Aeronautical Engineering, Sri Ramakrishna Engineering College, Tamil Nadu, India.

\*\*\*

**Abstract -:** Performance of a car is significantly affected by its aerodynamic properties. Aerodynamic properties must be considered for the purpose of studying the drag and stability performance of a car. In order to improve car aerodynamic drag and its stability, an aerodynamic device is needed such as spoiler. Rear spoiler is an aerodynamic device that functions to slow down and collect air. The objective of this study is to investigate the effects of spoiler on aerodynamic drag and stability of a car. Several spoilers design is attached at roof part of base line model. The base line car model with and without rear spoiler is designed with CATIA V5 and analyzed using ANSYS – FLUENT software to study different flow pattern happened over the car and thus the influence of aerodynamic drag could be predicted. From the result, albatross wing spoiler can significantly reduce drag, by creating high pressure at the roof.

**Key Words:** Spoiler, Aerodynamic Drag, Fluent

## I. INTRODUCTION

The performance and stability of an automobile is affected by its aerodynamic properties. Parts like rear spoilers, lower front and rear bumpers, air dams, etc., are directing the airflow in different way and results in greater drag reduction to the car and at the same time enhance the stability. Most of the automotive vehicles are designed with rear spoiler with an appropriate shape in order to reduce the aerodynamic drag and ground effect.

[1] During flight Albatross bird shows fine stability attitude irrespective speed, and gust load in the atmosphere. The shape of the airfoil of the bird wing plays major role for its best aerodynamic performance and stability. [2] Describes how to design the spoiler in CFD Software. [8], [9] This paper illustrates that by addition of components in the car model the lift and drag can be reduced. To increase down force rear spoiler is used for vehicle especially passenger car. It is an aerodynamic device that design to 'spoil' air movement across a car body. However spoiler also can be attached to roof or front/rear bumper as air dam. Some major aerodynamics factor which is lift and drag is contributed for rear spoiler. Fuel can be saved by reducing the drag force; To control stability spoiler is used.

## II. METHODOLOGY

A generic model of the passenger car with spoiler and without spoiler is prepared in the CATIA V5 software. ANSYS FLUENT tools was used to analyse the flow over a car and its coefficient of drag were significantly monitored.

## III. THE DESIGN

Most of the automotive car spoilers are designed with the airfoil shape which would produce high speed but results with notable increase in drag which would affect the overall performance of the car. Type of spoiler and those design of car model must be capable and suitable in order to increase its performance and stability factors during riding. Albatross wing have its unique features to meet the abovesaid performance factors. The profile of albatross wing is taken for spoiler model. The design of albatross wing is done in the CATIA V5 software and the spoiler is fixed to the car roof and is taken to analysis in ANSYS fluent.

## IV. CAR MODEL WITH SPOILER AND WITH OUT SPOILER

Modelling for the analysis were taken as with and without spoiler. 1:8 scaled down albatross spoiler model has been designed using CATIA V5 tool. Actual dimension of the car and the spoiler were scaled in order to maintain the conceptual flow phenomena over the large scale car model with and without the spoiler. The spoiler models made fix with the base line model. This precaution step can avoid any errors during analysis and also to interpret the flow phenomena.

The models are designed in two ways namely

- Car model with spoiler
- Car model without spoiler

Car Model is the Baseline model (BLM) for the analysis.

## V. GEOMETRY MESH

This surface mesh is tetrahedron type is generated on its surface. A surface mesh size 1.00mm

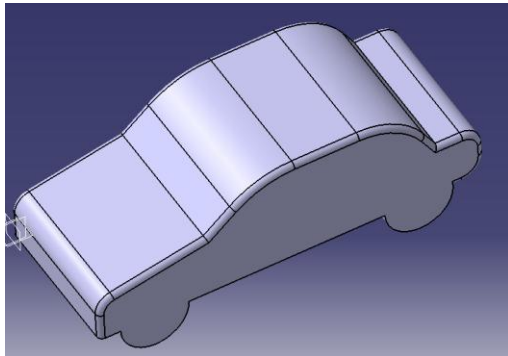


Figure 1: BLM without spoiler

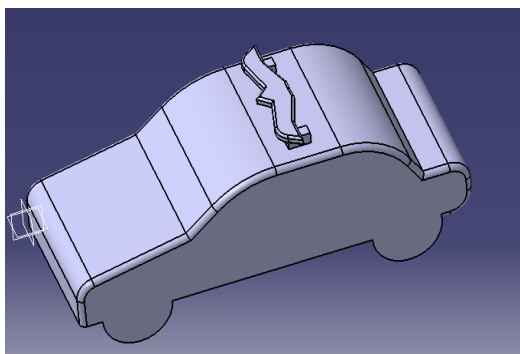


Figure 2: BLM with spoiler

### VI. ANALYSIS

ANSYS FLUENT 16.0 is used to analyse the car model with its attachment, which is the spoiler i.e albatross wing spoiler. FLUENT is the fluid flow analysis tool. The designed car model is allowed to undergo a flow over it by keeping the car model (solid) without spoiler within a fluid enclose in the analysis. Respectively the same is repeated for the model with albatross airfoil spoiler.

Besides that, some limitations must be considered during the analysis model to make sure that the results are acceptable i.e., Ground line must be 1mm. Ground line is a distance between road surface and bottom part of the car an, top line 260 ,side line is 75 on both side and front and back line must be 175. This distance is important to keep in constant so that the  $C_D$  and  $C_L$  are acceptable for designing the car. Another part that is important is the location of fixing the spoiler so as that the pressure acting at the roof is in same region.

### VII. BOUNDARY CONDITIONS

The set up were given with the proper boundary conditions to make the analysis. For the qualitative approach the flow is considered as inviscid.

### Input parameters

Velocity inlet	Magnitude	22m/s
	Turbulence Specification method	Intensity and viscosity ratio
	Turbulence Intensity	1.00%
Pressure outlet	Turbulence viscosity ratio	20
	Gauge pressure magnitude	0 Pascal
	Gauge pressure direction	Normal to boundary
	Turbulence specification method	Intensity and viscosity ratio
Wall zone	Backflow turbulence Intensity	10%
	Backflow turbulence viscosity ratio	10
	Car surface	no slip
Fluid properties	Ground face	no slip
	Side face	no slip
	Fluid type	Air
	Density	1.225

### VIII. RESULTS

The streamline patterns evident better understanding of the vortex phenomena happening in a flow. The velocity and pressure flow streamlines were considered as results to validate the flow pattern over the albatross profile. These patterns are related to interpret aerodynamic drag generated in the downstream of the car. Comparison of flow streamlines with and without spoilers are considered for better illustration of drag, aerodynamic lift and influences of spoiler.

#### Velocity streamline flow without spoiler

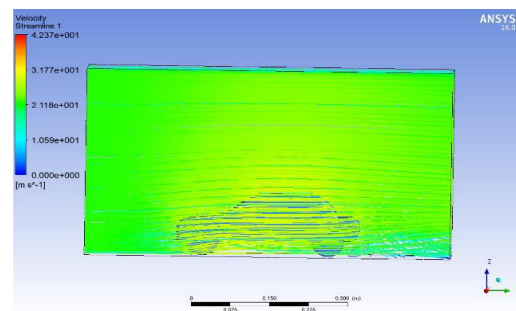


Figure 3 : Velocity streamline flow Side view

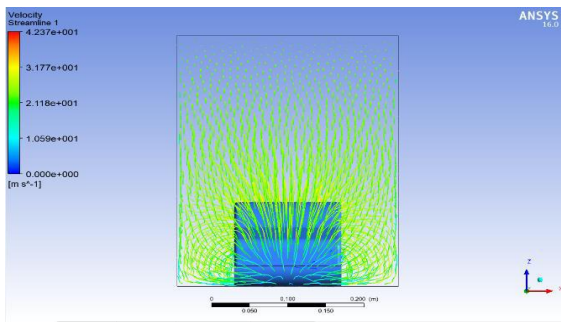


Figure 4: Velocity streamline flow Front view

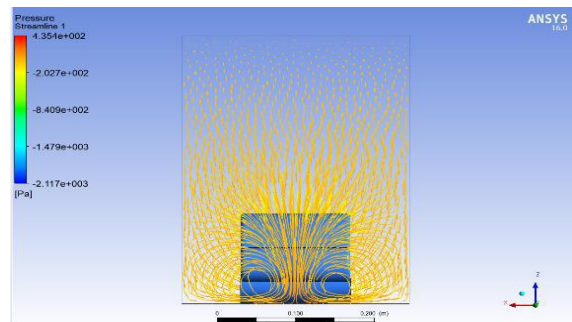


Figure 8: Pressure streamline flow Back view

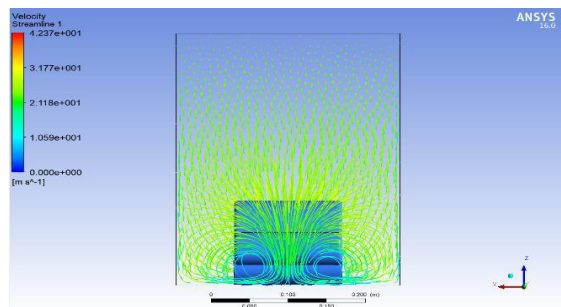


Figure 5: Velocity streamline flow Back view

**Velocity contour flow (without spoiler)**

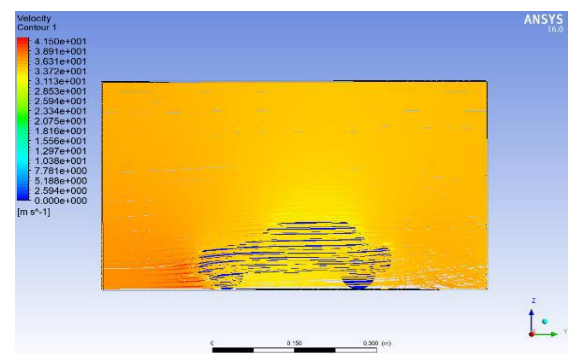


Figure 9: Velocity contour flow Side view

**Pressure streamline flow (without spoiler)**

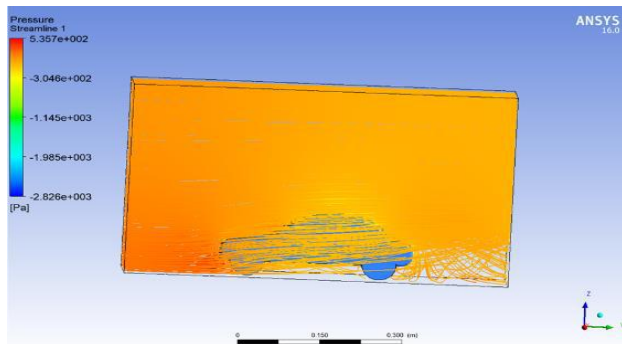


Figure 6: Pressure streamline flow Side view

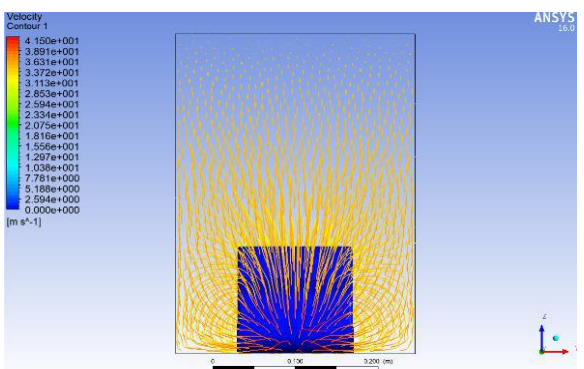


Figure 10: Velocity contour flow Front view

**Pressure contour flow (without spoiler)**

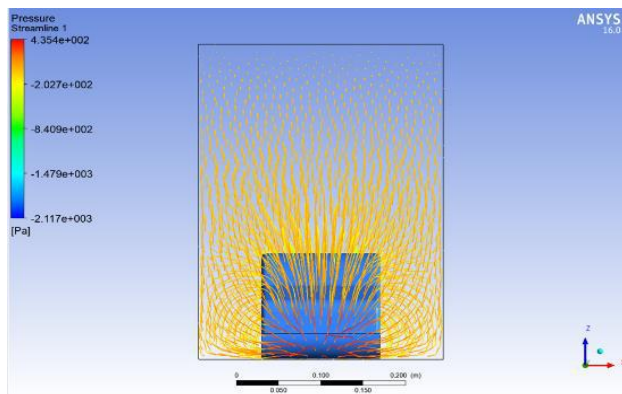


Figure 7: Pressure streamline flow front view

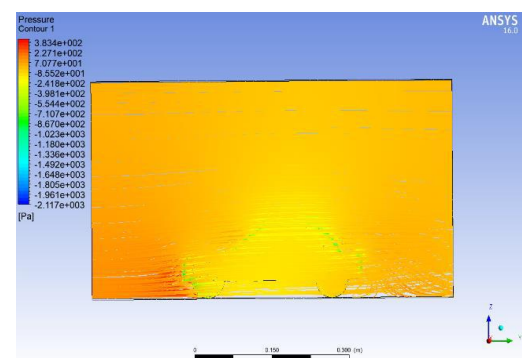


Figure 11: Pressure contour flow side view

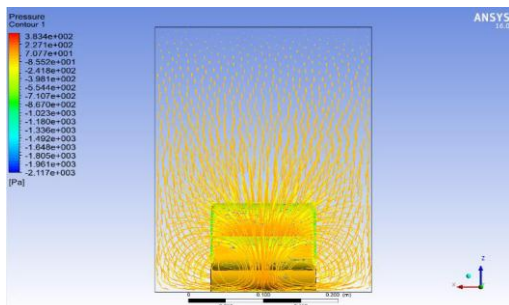


Figure 12: Pressure contour flow Back view

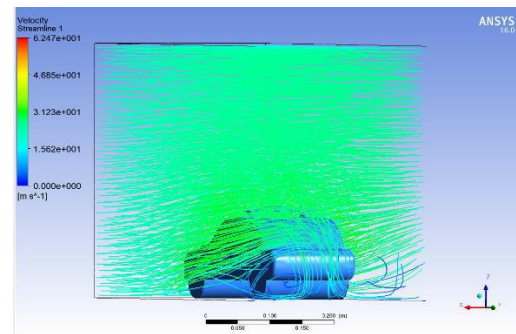


Figure 16: Velocity streamline flow Back view

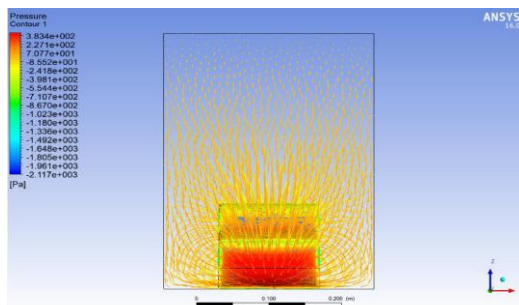


Figure 13: Pressure contour flow Front view

Pressure streamline flow(with spoiler)

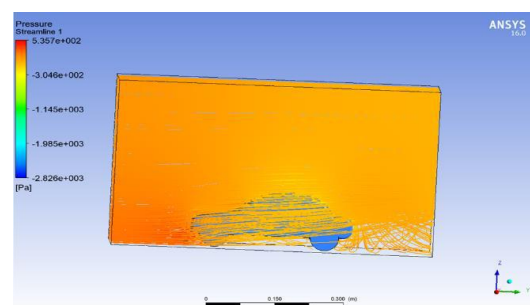


Figure 17: Pressure streamline flow Side view

Velocity streamline flow(with spoiler)

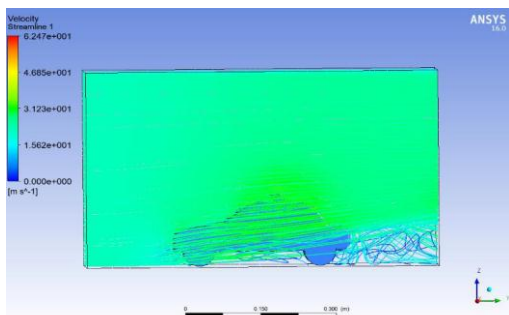


Figure 14: Velocity streamline flow Side view

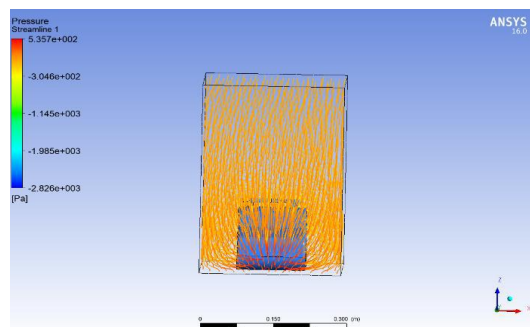


Figure 18: Pressure streamline flow Front view

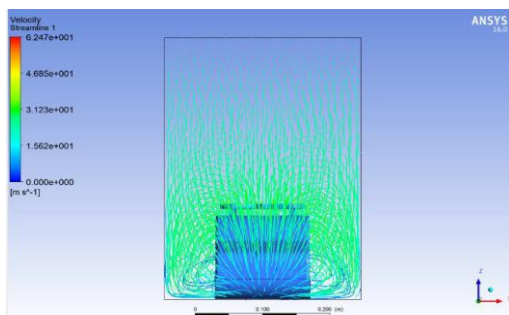


Figure 15: Velocity streamline flow Front view

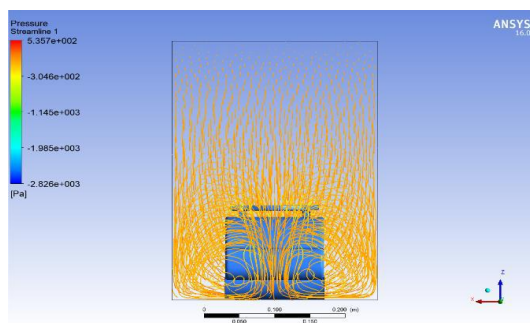


Figure 19: Pressure streamline flow Back view

**Velocity contour flow (with spoiler)**

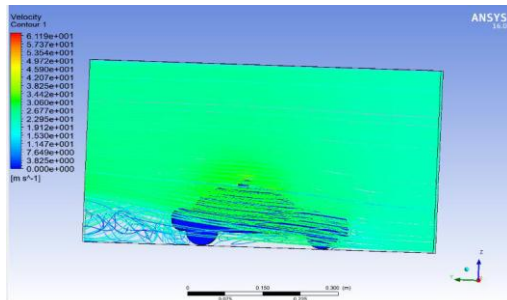


Figure 20: Velocity contour flow Side view

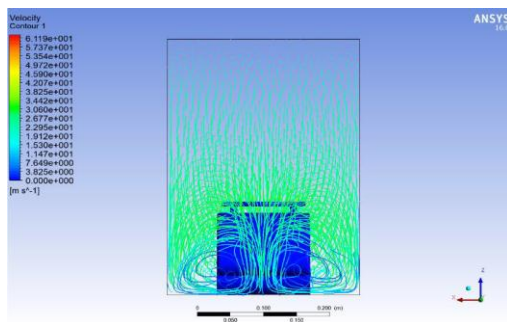


Figure 21: Velocity contour flow Back view

**Pressure contour flow (with spoiler)**

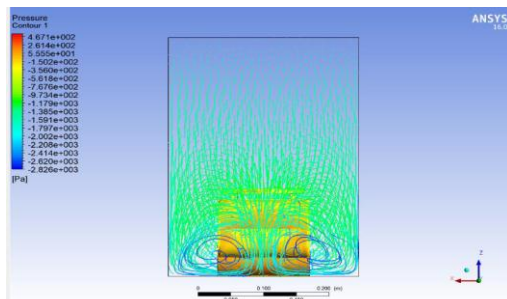


Figure 22: Pressure contour flow Back side

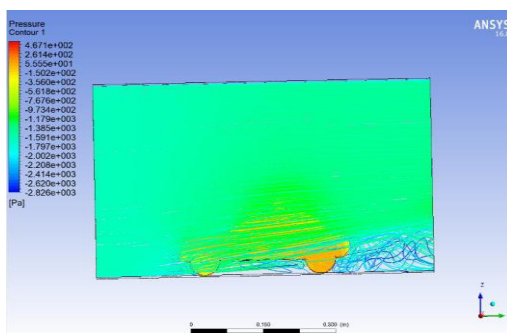


Figure 23 : Pressure contour flow Side view

**IX. CONCLUSIONS**

As the analysis results evident clear flow pattern downstream of the car with and without spoiler. Aerodynamic drag is significantly reduced as in the case of Albatross wing spoiler placed on the top of the car model. Apparently 19% of the flow speed is increased in the albatross wing profile rather than the car model with normal wing platform spoiler. It also results with 30 percent of speed increment and 24 percent of drag reduction than the car model without spoiler. Thus albatross wing profile spoiler shows better performance of a car.

**ACKNOWLEDGEMENT**

We thank the Department of Aeronautical Engineering of Sri Ramakrishna Engineering College for providing the facilities to complete the project successfully.

**REFERENCES**

- [1]. Günther Pfeifhofer, Helmut Tributsch, "The Flight of Albatross—How to Transform It into Aerodynamic Engineering?", Scientific research, 2014.
- [2]. Xu-Xia Hu, Eric T.T. Wong, " A numerical study on rear spoiler of passenger vehicle" Waset.Org publications , 2011
- [3]. Q.Jia, M.M. Rashidi, SB. Ju, ZG. Yang, "The influence of wheel spoiler on vehicle drag reduction", Science PG publications, 2015
- [4].Anantha Raman , Rahul Hari H , "Methods for reducing aerodynamic drag in vehicle and thus acquiring fuel economy, Journal of advanced engineering research, 2016.
- [5]. G. Siva , V.Loganathan, "Design and aerodynamic analysis of a car to improve performance",IDOSI publications , 2016
- [6].Ahmad Zainal Taufik, Nur Rashid , Mazlan, Mohd Faruq, Muhammad Zahir, "Electric car chassis design by using CATIA V5 R19",IOSR Publications , 2014
- [7]. Vikas .V Chaurasiya ,Deepak .B Kushwaha, Mohd-Raees, "Aerodynamic analysis of vehicle using CFD" , International journal of recent trends in engineering and research,2017
- [8]. E.M.Wahba, H. Al Marzooqi, M . Shaath , M . Shahin , "Aerodynamic drag reduction for ground vehicles using lateral guide", CFD letters ,2012.
- [9]. A . Anish, Suthen .P. G, Viju. M. K, "Modelling And Analysis Of A Car For Reducing Aerodynamic Force", 2017.
- [10]. Padmagari Naresh, N.Sreenivas, "Design And Analysis Of A Car Spoiler For Drag Reduction", edupedia publication , 2017.