Computational Aerodynamic Analysis of a Rear Spoiler on a Car in Two Dimensions

Parag S Kulkarni[1] & Prof. R S Shelke[1]

[1]Program Manager, Department-Program Management, Flex GBS, Pune, India
E-mail: kparag575@gmail.com

[2]Associate Professor, Department-Mechanical Engineering, S.V.I.T COE, Pune, India
E-mail: rssme@rediffmail.com

Abstract: Performance, safety, maneuverability of a car depends on multi-disciplinary elements/ factors such as car engine, tyre, aerodynamics, and ergonomics of design and most proficiently the driver. Improvement in the aerodynamic drag can be achieved in multiple ways of introducing active and passive air flow control. Rear spoilers are an example of the passive air flow control of the aerodynamic drag. Generally rear spoilers are used to slower down the air flow and accumulate air which helps increasing the pressure around the trunk and removing any chance of low pressure. The research investigates on the effect of the rear spoiler in the aerodynamic drag, stability and efficiency. The research focuses on 2D model of BMW 3 series sedan car with & without spoilers and the iterations of the rear spoilers are designed in Auto desk inventor software. Modifications in the rear spoilers are done to obtain the minimal drag and maximum downward force. The use of CFD software is to calculate the estimated drag and lift values acting on the car as well as the drag force and the coefficient of lift to improve the drag & stability. It involves understanding the basic applications of the post processing tools. The results showed that the rear spoilers help in reducing drag by creating high pressure at the rear of the car.

Keywords: Manoeuvrability, Aerodynamics, Air flow, Spoilers, CFD.

I. INTRODUCTION

Aerodynamics for the cars has changed gradually from initial designers to the manufacturers to obtain more power under the hood. This means more stability; better performance, better grip and most prominently increase the comfort of the car. People seem to have sportier look to have the best output performance. This certainly does mean that the cars are equipped with more additional parts such as air dams, front and rear spoilers, and use of VGs (vortex generators) on the surface of the cars. Most widely used are the rear spoilers in the passenger cars. This aids in greater drag reduction and in the same occasion increases the stability of the car.

Mostly mounted on the car's rear depending on the fixing location of the car rear either a fastback, notch-back or square back. Spoilers can even be mounted in the front of the car as air dams with the bumpers. However, rear spoilers provide the maximum contribution to the aerodynamic drag and lift. This occurs as rear spoilers stagnant the flow of the air at the rear of the car generating a high-pressure region and reducing the low pressure. This directs the flow and offer greater drag reduction, increasing the downward force at the rear and more stability.

Usually when a person drives the car, the car breaks through the barrier of the air. This creates a region of high pressure as the air flows from the windscreen to the top surface of the car. Gradually there is a region of the low pressure created at the rear of the car. In a worst-case scenario, the air which possibly makes way to the rear window creates a notch due to the window dropping down to the trunk, creates a region of vacuum or low pressure which lifts the car and acts on the surface area of the trunk. This is possibly because of the lack of the air being refilled in that region.

Technically a spoiler regulates the flow of air around the rear end by accumulating more air refill in the region of the low pressure so that more high-pressure region is created with better stability and the car always sticks to the ground. Use of spoiler is quite unique and impressive as most of the sedan & hatch back cars tend to become light at the rear end lifts the car while the spoilers help acting as an air barrier. This also allows reducing the axle-lift and reduction of dirt in the rear surfaces of the car.

II. RESEARCH AIM AND OBJECTIVES

The research project aims to accumulate all possible information & Knowledge of a model car BMW 3 series sedan class aerodynamics focusing on the rear spoiler use. Aerodynamic forces can be used to improve the tyre adhesive nature and find the vehicle performance. It describes the side slipping forces acting on the tyre. Using three different types of the rear spoilers & their CFD analysis results to achieve the aim using following objectives in the research project.

1. Analysis of the air flow around the car without the rear spoiler.
2. Analysis of the air flow around the car with a concept rear spoiler.
3. Effect of the aerodynamics on the car
4. Analysis with the variation of the rear spoilers on the down force, drag and the stability of the car.
5. Estimating the CD (Coefficient of Drift) & CL (Coefficient of Lift) on a high-speed run of the car.
6. Comparison of the CD and CL values with (variations in rear spoiler designs) and without rear spoilers.
7. Analysis of all the models on the CFD software ANSYS Fluent.
8. Drawing out the possible outcomes comparing the results & establishing the relation of using rear spoilers for better performance, reduced lift and drag.

III. METHODOLOGY PREPARE YOUR PAPER BEFORE STYLING

A. Computational Fluid Dynamics

Fluids (gasses and liquid) are governed by partial equations that represent the general laws of conservation of mass, momentum and energy. CFD is the art of replacing such PDE by set of equations which can be solved by the digital computers.

Computational Fluid Dynamics (CFD) provides quantitative and qualitative predictions of the fluid flow by means of the following:

- Modelling by applications of mathematics of partial differential equations
- Use of discretion and solution tools i.e. numerical methods.
- Use of the software tools like solvers, pre and post processing utilities.

CFD is essential software which enables the engineers to virtually simulate the numerical experiments carried in the laboratories resulting in less time-consuming process and better accurate results. CFD gives an insight to the pattern of the fluid flow that is difficult to predict with regular experiments, expensive to conduct and sometimes impossible to study by the regular experiments.

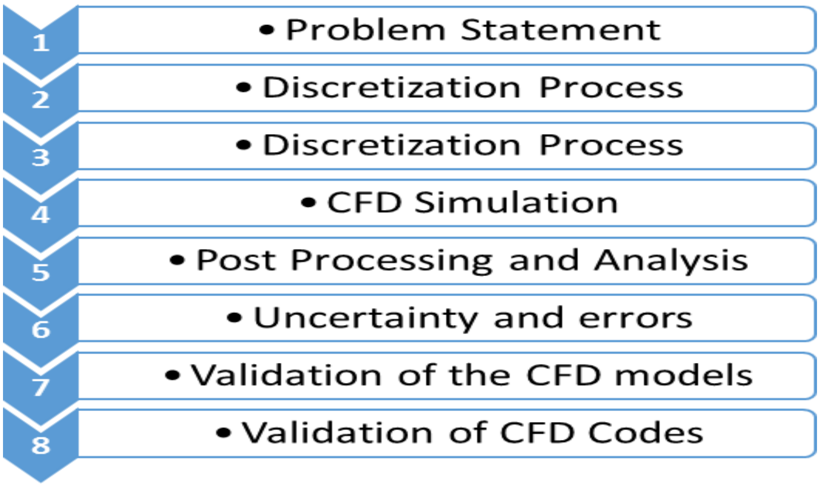

Figure 1. Procedure followed in CFD Analysis

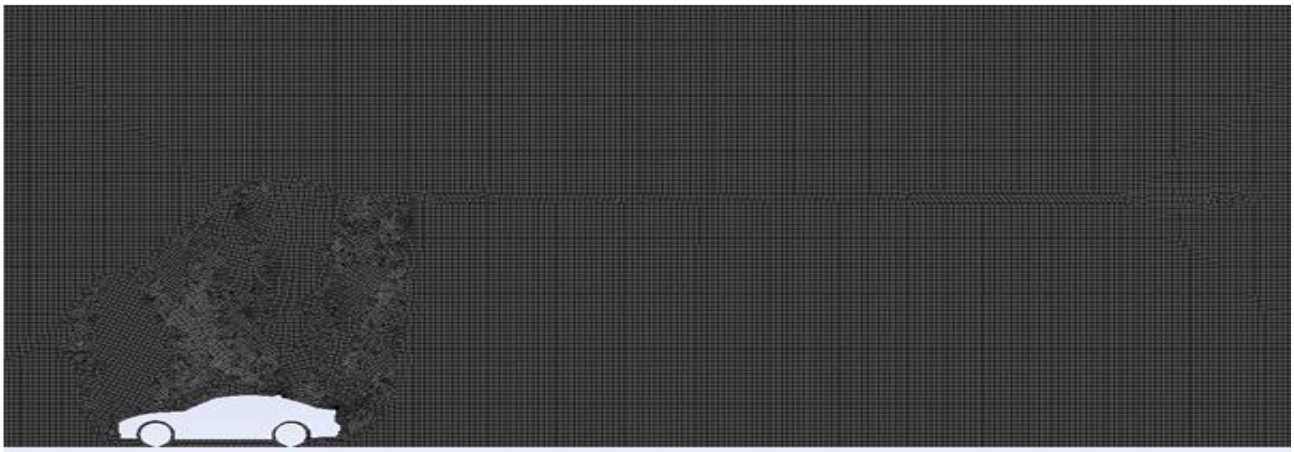

Figure 2. Generation of unstructured mesh of BMW 3 series model.

IV. ANSYS FLUENT RESULTS & ANALYSIS

Every model is run on the ANSYS Fluent to obtain a simulated picture which shows different physical properties that affect the model. The colored picture depicts the values of the physical properties at the instant. This half of the chapter is to analyze the models with the results obtained in the pictorial form and find the coherence with the graphs and physical theories.

A. Analysis For Blm

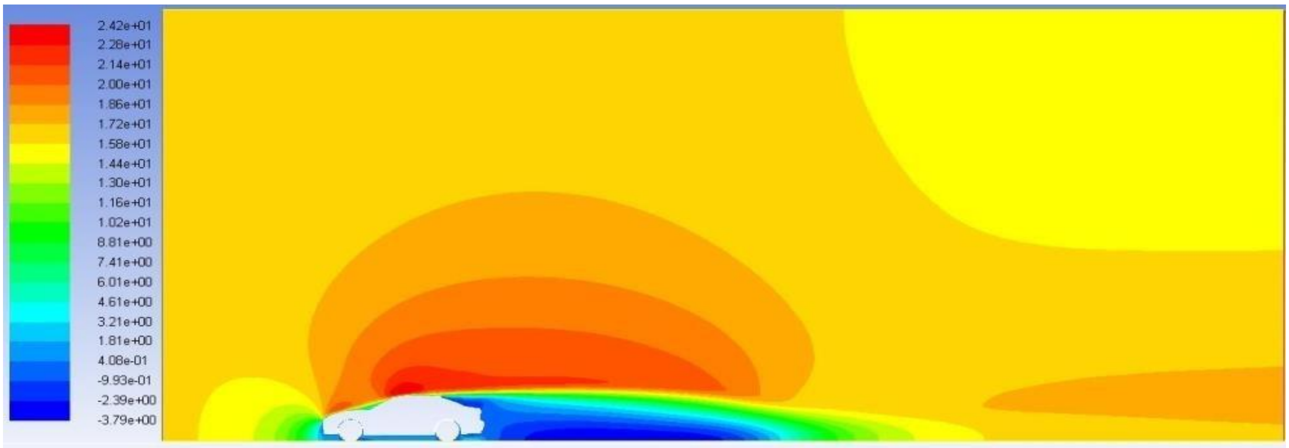

Figure 3. Velocity magnitude picture from Fluent

B. Analysis For Manufacturer Model

i. Velocity Contours:

As explained in the model car without any spoiler, the figure 84 below shows the velocity magnitude of the model with built-in spoiler. The magnitude of velocity of the air as it hits the car bonnet top surface starts to increase from 16.40 m/s and reaches at a maximum of 24 m/s or greater on the surface of the bonnet. Similar phenomenon happens at the top surface of the model car roof. Further moving with the top geometry of the model car to the rear part the velocity approaches to 0.

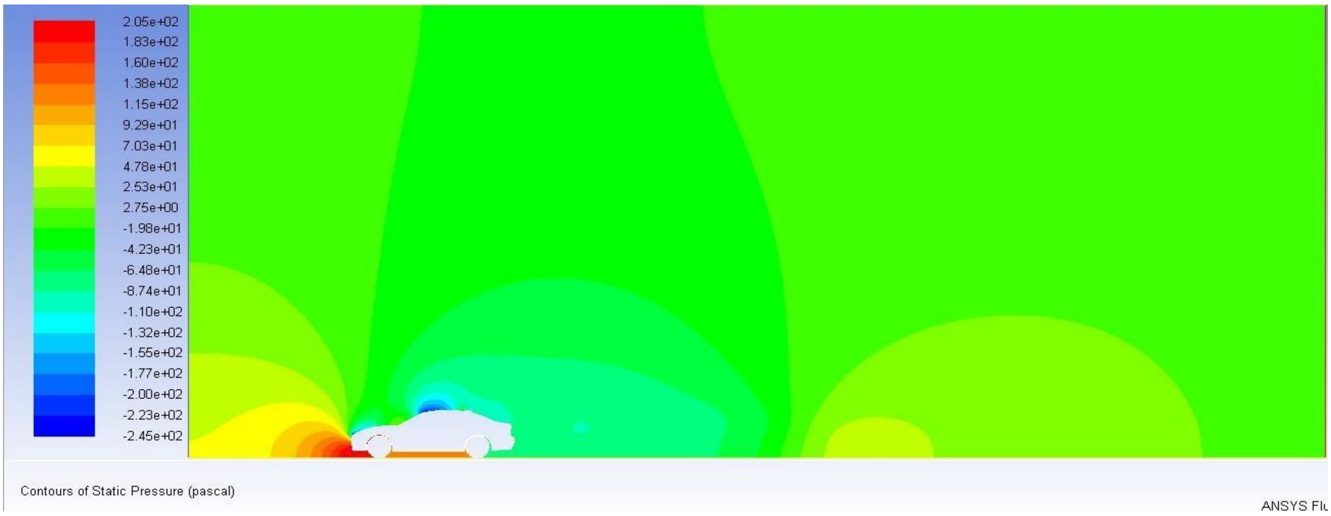

Figure 4. Velocity magnitude in manufacturer's –built in model

C. Turbulence Comparison table:

Model	*Turbulence region/ value*	*Description*
BLM – Normal car	The region of turbulence is way behind the car.	The turbulence does not have or matches with the streamline flow. The wake region is quiet near to the car rear field.
Built-in spoiler model	The region of the turbulence is not very far from the model.	The wake region and the area of the recirculation lies very close approximately twice the distance of the car.
Deck-lid Spoiler model	The turbulent region is quite widespread almost very far	The turbulent region shows a stream line recirculation. The wake is widespread but obviously stream lined.

IV. CONCLUSION

Different types of cars in sedan class use rear trunk spoilers. The research focused on the application of the rear spoiler designed in 2D and explained in CFD processing tools which address the problem statement by allowing the model car ‘BMW 3 series to withdraw its drag and reducing the lift. This has the effect of streamlining the model car to attain the lowest possible drag and lift when required in high velocity.

The research focused generalizing the basic concepts of the governing laws and theories of the fluid flow around the spoilers. This helped in establishing the concept with the product. Apart giving a broad introspect to the concepts, this was followed by the research methodology which dealt with the basic approach of the research work. Being quantitative in nature and mixture of qualitative techniques resulted in the scope of understanding the different ways of approaching the problem statement. The resultant comparison of the drag forces and lift forces well establish that a normal car without the rear spoiler has higher values than that of the drag and lift compared to the model with the spoilers. Compared to other models, the built-in model provided the best results for the aerodynamic forces.

Conflict of interest: The authors declare that they have no conflict of interest.

Ethical statement: The authors declare that they have followed ethical responsibilities

REFERENCES

[1] Aberu, R. (2013). Adaptrive Automobile Aerodynamics. Johannesburg: University of Johannesburg.

[2] Altecc.(2001-2015). Cartecc.Retrieved March 15,2015,from http://www.kfztech.de/Engl/Hersteller/Alfa

[3] Ann M. Anderson and Jeremy Losaw. (2016). Using Race Car Aerodynamics to Teach Mechanical Engineering. Schenectady, NY, USA: American Society for Engineering Education.

[4] Autodesk Inventor Professional. (2014, March Tuesday). Autodesk Inventor Help . Retrieved April 16, 2015, from Autodesk Inventor: http://help.autodesk.com/view/INVNTOR/2014/ENU/?guid=GUID-B1667D21-A38F-4B20-901B-186EA039DF5A

[5] Bakker, A. (2015). Bakker Organisation. Retrieved April 26, 2015, from Computational Fluid Dynamics: http://www.bakker.org/dartmouth06/engs150/06-bound.pdf

[6] BMW UK. (2017, April 13). BMW 3 series techinical data. Retrieved April Tuesday, 2017, from BMW UK: http://www.bmw.co.uk/en_GB/new-vehicles/3/saloon/2017/technical-data.html

[7] Computational Drag Analysis of Passenger Car Using Splines and Spoiler. (2015). International Journal of Engineering Trends and Technology, 1-11.
[8] Daniel;. (2012). Aerodynamic analysis and optimisation of the rear wing of a WRC car. Oxford: Oxford Brookes University, School of Technology.
[9] Dimitriadis, G. (2014). Vehicle Aerodynamics. SAE International (pp. 33-40). Italy: University de Liege.
[10] Dimitris, G. (2017). Vechile Aerodynamics. Retrieved April 26, 2017, from Experimental Aerodynamics: http://www.ltas-aea.ulg.ac.be/cms/uploads/VehicleAerodynamics02.pdf
[11] Duysinx, P. (2014-2015). Performance of vehicles. Performance of vehicles (pp. 1-70). Belgium: University of Liege.

This volume is dedicated to Late Sh. Ram Singh Phanden, father of Dr. Rakesh Kumar Phanden.

