

Investigation of Drag and Lift Forces over the Profile of Car with Rears spoiler using CFD

V. Naveen Kumar^{*}, K. Lalit Narayan, L. N. V. Narasimha Rao and Y. Sri Ram

Mechanical Engineering, Sir. C.R. Reddy College of Engineering, Andhra Pradesh, India

*Correspondence Info:

V. Naveen Kumar

Mechanical Engineering,

Sir. C.R. Reddy College of Engineering, Andhra Pradesh, India

E-mail: naveenmech524@gmail.com

Abstract

Now a days demand of a high speed car is increasing in which vehicle stability is of major concern. Forces like drag& lift, weight, side forces and thrust acts on a vehicle when moving on road which significantly affect the fuel consumption The drag force is produced by relative motion between air and vehicle and about 60% of total drag is produced at the rear end. Reduction of drag force at the rear end improves the fuel utilization. This work aims to reduce the drag force which improves fuel utilization and protects environment as well. In the stage of work a sedan car with different types of spoilers are used to reduce the aerodynamic drag force. The design of sedan car has been done on CATIA-2010 and the same is used for analysis in ANSYS-(fluent). The analysis is done for finding out drag and lift forces at different velocities, and spoilers. This study proposes an effective numerical model based on the computational fluid dynamics (CFD) approach to obtain the flow structure around a passenger car with a rear spoiler

Keywords: Car, Rears spoiler, CFD.

1. Introduction

Nowadays the everyday cars are changed by their owners to make the look sportier. Having more power under the hood leads to higher speeds for which the aerodynamic properties of the car given by the designer are not enough to offer the required down force and handling. The performance, handling, safety, and comfort of an automobile are significantly affected by its aerodynamic properties. Extra parts are added to the body like rear spoilers, lower front and rear bumpers, air dams and many more aerodynamics aids as to direct the airflow in different way and offer greater drag reduction to the car and at the same time enhance the stability. In case of that, many aerodynamics aids are sold in market mostly rear spoiler. Rear spoiler is a component to increase down force for vehicle especially passenger car. It is an aerodynamic device that design to 'spoil' unfavourable air movement across a car body. Main fixing location is at rear portion, depends on shape of the rear portion either the car is square back, notchback or fastback because not all rear spoiler can be fix at any type of rear portion of a car. However spoiler also can be attached to front rear bumper as air dam. Rear spoiler contributed some major aerodynamics factor which is

lift and drag. The reduction of drag force can save fuel; moreover spoiler also can be used to control stability at cornering. Besides can reduce drag and reduce rear-axle lift, rear spoiler also can reduce dirt on the rear surface.



Figure 1: The rear spoiler

1.1. What is Aerodynamics?

Aerodynamics is the way objects move through air. The rules of aerodynamics explain how an airplane is able to fly. Anything that moves through air is affected by aerodynamics, from a rocket blasting off, to a kite flying. Since they are surrounded by air, even cars are affected by aerodynamics.

“Aerodynamics” is a branch of fluid dynamics concerned with studying the motion of air, particularly when it interacts with a moving object

1.2. What IS CFD?

“CFD (Computational fluid dynamics) is a set of numerical methods applied to obtain approximate solution of problems of fluid dynamics and heat transfer.”

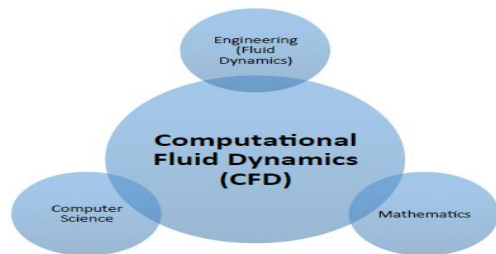


Figure 2: The different disciplines contained within computational fluid dynamics

According to this definition, CFD is not a science by itself but a way to apply methods of one discipline (numerical analysis) to another (heat and mass transfer). In retrospect, it is integrating not only the disciplines of fluid mechanics with mathematics but also with computer science as illustrated in **Figure 2**. The physical characteristics of the fluid motion can usually be described through fundamental mathematical equations, usually in partial differential form, which govern a process of interest and are often called governing equations in CFD. Jiyuan Tu, Guan Heng Yeoh and Chaoqun Liu [2] have discussed how to solve mathematical equations with using CFD:

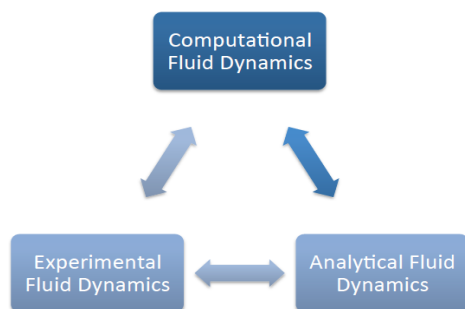


Figure 3: The three basic approaches to solve problems in fluid dynamics and heat transfer.

CFD has also become one of the three basic methods or approaches that can be employed to solve problems in fluid dynamics and heat transfer. As demonstrated in **Figure 3**, each approach is strongly interlinked and does not lie in isolation.

2. Problem Statement

When a driver drives his or her car in high speed condition, especially at highway which is speed limit 110 km/h, the car has high tendency to lift over. This is possible to happen because as the higher pressure air in front of the windshield travels over the

windshield; it accelerates, causing the pressure to drop. This lower pressure literally lifts on the car’s roof as the air passes over it. Worse still, once the air makes its way to rear window, the notch created by the window dropping down to the trunk leaves a vacuum or lower pressure space that the air is not able to fill properly. The flow is said to detach and the resulting lower pressure creates lift that then acts upon the surface area of the trunk. To reduce lift that acted on the rear trunk, a rear spoiler can attach on it to create more high pressure. Spoilers are used primarily on sedan-type cars. They act like barriers to air flow, in order to build up higher air pressure in front of the spoiler. This is useful, because as mentioned previously, a sedan car tends to become "Light" in the rear end as the low pressure area above the trunk lifts the rear end of the car.

3. Vehicle generic models and dimensions

The below Figure 4, outlines the steps before the lift and drag coefficient of New manza is obtained. The coefficient of lift is obtained for the front and rear section of the car. Overall surface of the new manza is scanned and exported to CAD modeling software (CATIA) to smooth and repair the surface. The CFD analysis is done using Star-CFD software. For the initial run, the result of *C_d* is compared to the standard published new manza value of *C_d* which is around 0.32. If the initial run is far away from the standard value, then the CAD model is checked and repaired again until the *C_d* value comes within 0.3 > *C_d* > 0.4 range. The values of drag and pressure coefficient is run at least 5 times and averaged. It is important to average this value to ensure the result is consistent and acceptable. The post-processor plot the data of pressure distribution and velocity vector on the centre section of the car. The center section of the car is the ideal value since it represents the closest outline to the real shape of the car. Then, the target is to analyze the pressure distribution and velocity vectors pattern that effect the drag, lift or down force. Finally to ensure the vehicle achieve the occurrence of down forces at both the front and the rear side of the vehicle which ensure good vehicle stability.



Figure 4: Dimensions of the generic vehicle model [Side-view]



Figure 5: Dimensions of the generic vehicle model [frant-view]

4. Spoiler generic models and dimensions

In the numerical analyze, two different spoiler styles have been used. The first spoiler was a “wing” style spoiler, which was mounted 28 cm above the surface of the vehicle’s rear-end, on the other hand the second spoiler was mounted edge of the rear side of the vehicle without leaving a gap between spoiler and the surface of vehicle. The generic model of the first spoilers is shown in Figure 6, below, while the generic model of second spoilers is shown in Figure 7, with relevant dimensions.

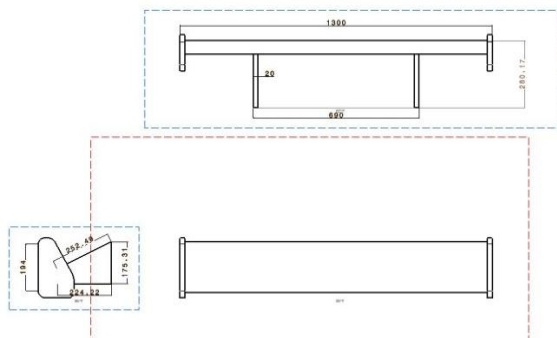


Figure 6: Generic model and dimensions of first spoiler

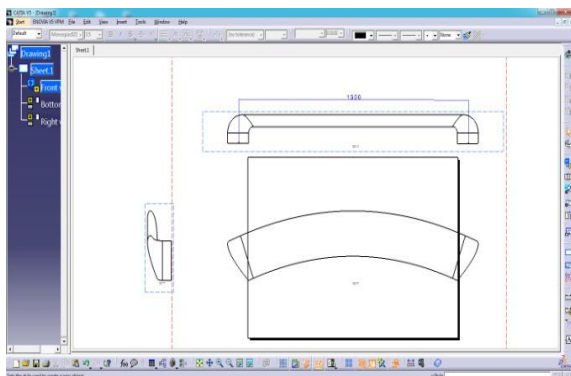


Figure 7. Generic model and dimensions of second spoiler

The vehicle itself (Figure 8.), vehicle with first spoiler (Figure 9.) and vehicle with second spoiler (Figure 10.) 3D CAD models shown above have been orientated in the virtual wind tunnel one-by one to performed three cases, benchmark #1, benchmark #2 and benchmark #3.

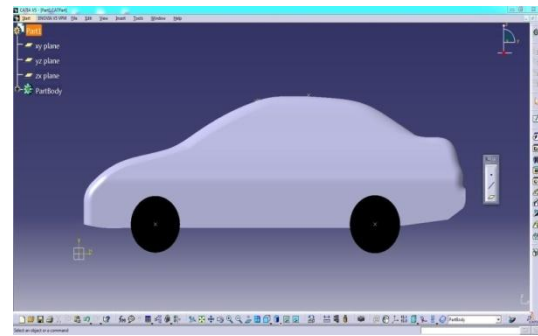


Figure 8: Vehicle 3D CAD model

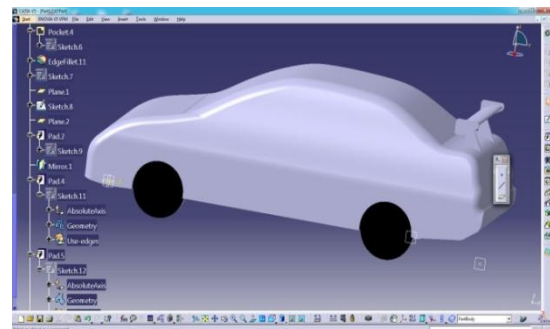


Figure 9: Assembly 3D CAD model of vehicle and first spoiler

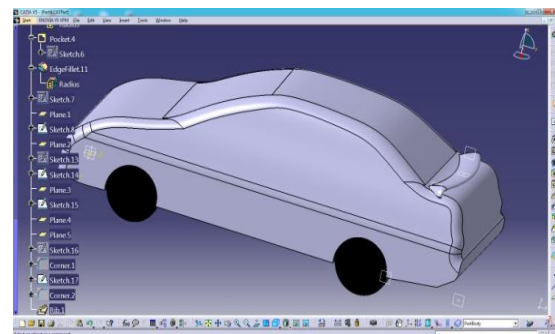


Figure 10: Assembly 3D CAD model of vehicle and second spoiler

5. Virtual wind tunnel and vehicle orientation

A virtual air-box has been created around the 3D CAD model (Figure 11.), which represents the wind tunnel in the real life. Since we are more interested in the rear side of vehicle, which is where the “wake of vehicle” phenomenon occurs, more space has been left in the rear side of the vehicle model to capture the flow behavior mostly behind the vehicle.

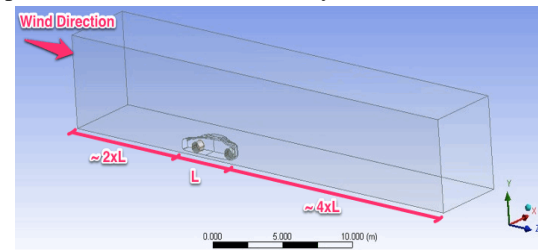


Figure11: Virtual wind tunnel and the vehicle orientation

Due to the complexity of the simulation with limited computer resources and time, the complete domain was divided to half using a symmetry plane (YZ plane), which means, the simulation would be calculated for just the one side of the vehicle and since the other side is symmetric and YZ plane has been defined as symmetric boundary in the solver to make the boundary condition as “a slip wall with zero shear forces”; the simulation results would be valid for full model as well. All 5 surfaces of the virtual wind tunnel (air-box) have been named so the numerical solver of ANSYS FLUENT® would recognize them and apply the appropriate boundary conditions automatically. The final meshing can be seen in **Figure 12**. The same procedure to create high resolution meshing has been followed for all cases (Case #1: Vehicle itself, Case #2: Vehicle with first spoiler, Case #3: Vehicle with second spoiler) exactly the same.

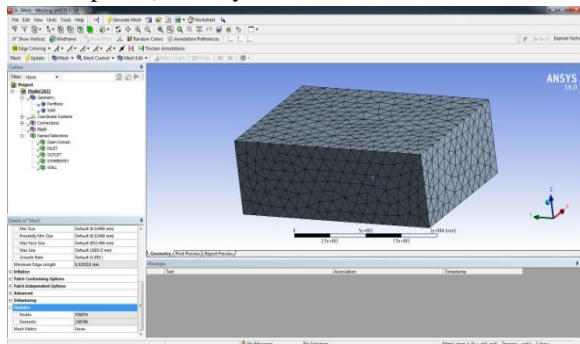


Figure 12: The final mesh

Table1: 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

6. CFD Analysis

Computational fluid dynamics (CFD) is an engineering method for simulating the behaviour of systems, processes and equipment involving flow of gases and liquids, heat and mass transfer, chemical reactions and related physical phenomena. More specifically CFD, or fluid simulation, can be used to reduce pressure drops, to predict aerodynamic lift or drag, to predict the rotor thrust, to calculate the airflow in air conditioned rooms, to ensure adequate cooling, to optimize mixing rates, and so on .ANSYS combines the most respected names in fluid simulation — ANSYS ® FLUENT® and ANSYS® CFX® — to expertly address your evolving CFD needs at a time when product reliability, safety and market performance are paramount. ANSYS offers the most complete suite of advanced CFD software tools

available, coupled with unrivaled modelling capabilities, to help you achieve a faster total time to solution. More product development leaders worldwide trust ANSYS software as their fluid dynamics simulation platform for its accuracy, reliability and speed. ANSYS is committed to providing world-class high-fidelity fluid dynamics technology for Simulation Driven Product Development. In addition to providing the most well validated and used CFD products on the market its ANSYS® Workbench™ platform is built on the ability to co-simulate. Users of ANSYS’ CFD solutions can include structural mechanics or electrical aspects to a model through our other industry-leading solver solutions. By that means customers reduce engineering assumptions and increase the fidelity of their models.CFD solutions from ANSYS deliver unprecedented productivity to help you analyze multiple, automated parametric design variations all without complex programming. ANSYS provides integrated tools that not only help you to conveniently understand which parameters your design is most sensitive to, but also determine which design parameters require the tightest control. Integrated tools for design (shape) optimization and Six Sigma analysis are also available. ANSYS simulation software gives you the power to design for robust performance and deliver better products, faster. CFD solutions from ANSYS are readily customizable and extensible to meet your ongoing simulation and workflow process requirements - whether it’s extending solver capabilities to predict unique environmental factors or customizing our products to automate your design work flow. With ANSYS tools, your designers have the power to create better products more profitably. Because they can yield significant benefits (incl. more innovation, cost savings, reduced development time, increased quality), they have become an integral part of the engineering design and analysis environment of companies in the widest range of industries.

6.1. Importance of computational fluid dynamics

There are three methods in study of Fluid: theory analysis, experiment and simulation (CFD). As a new method, CFD has many advantages compared to experiments. Please refer Table2

Table 2: Comparison of Simulation and Experiment

	Simulation (CFD)	Experiment
Cost	Cheap	Expensive
Time	Short	Long
Scale	Any	Small/Middle
Information	All	Measured point
Repeatable	Yes	Some
Safety	Yes	Some Dangerous

6.2. CFD Analysis without spoiler

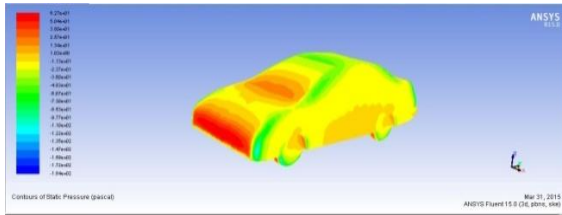


Figure 13(a): Static pressures

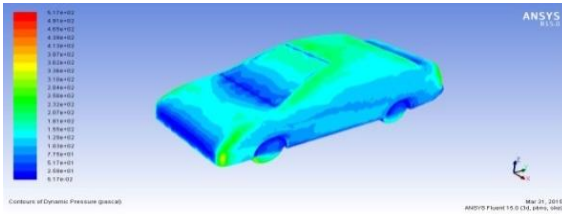


Figure 13(b): Dynamic pressure

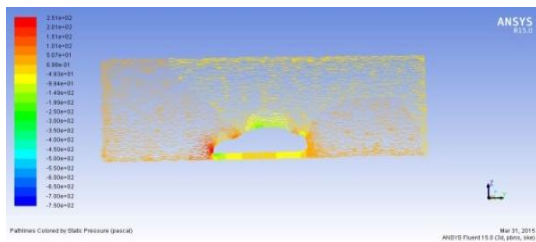


Figure 13(c): Pathlines by static pressure

The figure 13(a,b,c) represents the static and dynamic pressure of the vehicle. It is observed that the maximum pressure is 75.5 Pascal in front of the vehicle at bottom of the wind shield and in front of hood because of sharp edges

6.3. Velocity analysis

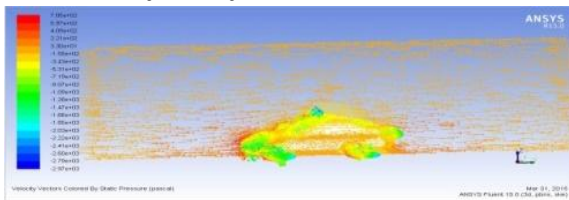


Figure 14(a). Velocity and magnitude [70km/hr]

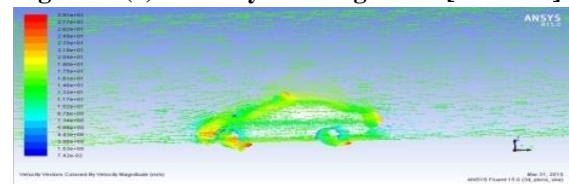


Figure 14(b). Velocity and magnitude [90km/hr]

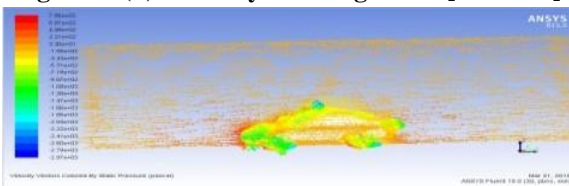


Figure 14(c). Velocity and magnitude [110km/hr]

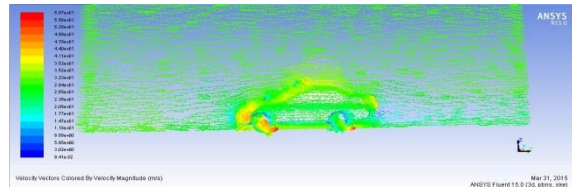


Figure 14(d). Velocity and magnitude [130km/hr]

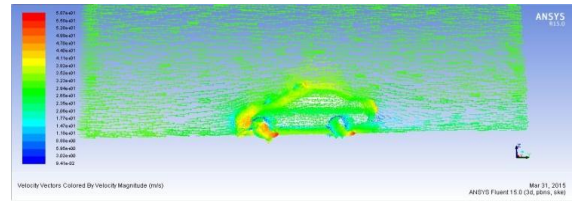


Figure 14(e). Velocity and magnitude [150km/hr]

Figure 14(a,b,c,d,e). represents CFD analysis without spoiler on different velocities [70, 90,110,130,150 km/hr]

6.4 CFD Analysis (Spoiler1)

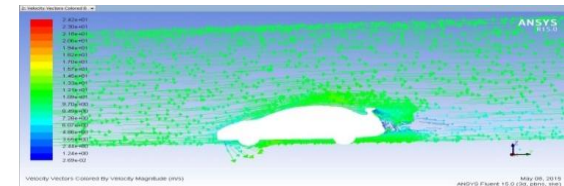


Figure 15(a): Velocity and magnitude [70km/hr]

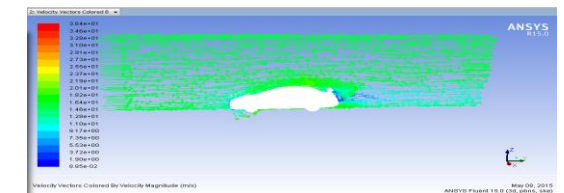


Figure 15(b): Velocity and magnitude [90km/hr]



Figure 15 (c): Velocity and magnitude [110km/hr]

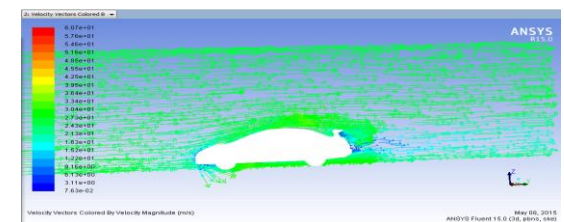


Figure 15 (d): Velocity and magnitude [130km/hr]

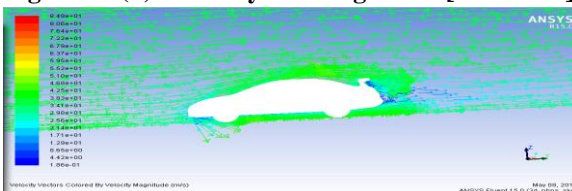


Figure 15 (e). Velocity and magnitude [150km/hr]

Figure 15(a,b,c,d,e).represents CFD analysis with spoiler1 on different velocities [70, 90,110,130,150 km/hr]

6.5 CFD Analysis (Spoiler2)

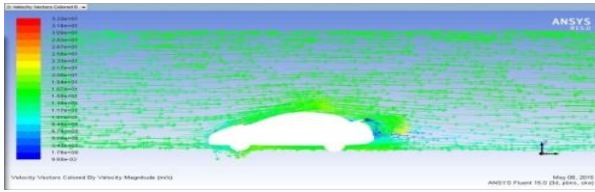


Figure 16(a): Velocity and magnitude [70km/hr]

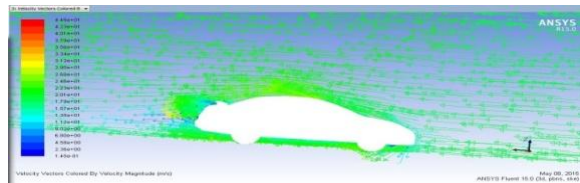


Figure 16(b): Velocity and magnitude [90km/hr]

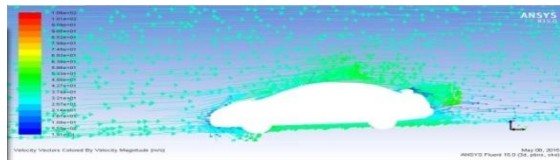


Figure 16(c). Velocity and magnitude [110km/hr]

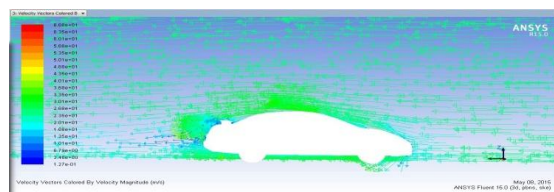


Figure 16(d): Velocity and magnitude [130km/hr]

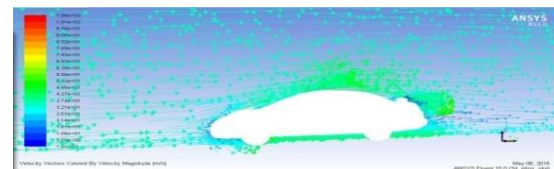


Figure 16(e). Velocity and magnitude [150km/hr]

Figure 16(a,b,c,d,e) represents CFD analysis with spoiler2 on different velocities [70, 90,110,130,150 km/hr]

7. Results and Discussion

The purpose of this chapter is to provide a review of past research efforts related to car aerodynamic drag & lift and its attachment for aerodynamics aids which is rear spoiler. A review of other relevant research studies is also provided. Substantial literature has been studied on aerodynamic drag, aerodynamic lift and influences from both and purpose of rear spoiler as one of the aerodynamic aids. The review is organized chronologically to offer insight to how past research efforts have laid the groundwork for subsequent studies, including the present research effort. The review is detailed so that the present research effort can be properly tailored to

add to the present body of literature as well as to justify the scope and direction of the present research effort

7.1 Data Presentation

Table 6.1: Car without Rear Spoiler

S. No	Velocity (Km/h)	Drag Force (N)	Lift Force (N)	Cd	Cl
1	70	153.86	51.109	0.329	0.1183
2	90	255.0	85.711	0.330	0.117
3	110	383.397	129.588	0.331	0.1153
4	130	536.83	182.00	0.333	0.108
5	150	718.946	248.474	0.334	0.096

Table 6.2: Car with Rear Spoiler 1

S. No	Velocity (Km/h)	Drag Force (N)	Lift Force (N)	Cd	Cl
1	70	147.431	49.646	0.3326	0.112
2	90	244.117	80.6639	0.329	0.110
3	110	396.157	125.973	0.331	0.115
4	130	512.359	172.825	0.330	0.113
5	150	684.323	224.034	0.333	0.110

Table 6.3: Car with Rear Spoiler 2

S. No	Velocity (Km/h)	Drag Force (N)	Lift Force (N)	Cd	Cl
1	70	144.062	49.867	0.325	0.1125
2	90	241.185	82.839	0.329	0.1130
3	110	364.775	127.397	0.333	0.1163
4	130	503.183	168.237	0.329	0.1100
5	150	670.066	216.906	0.329	0.1060

16. Graphical representation

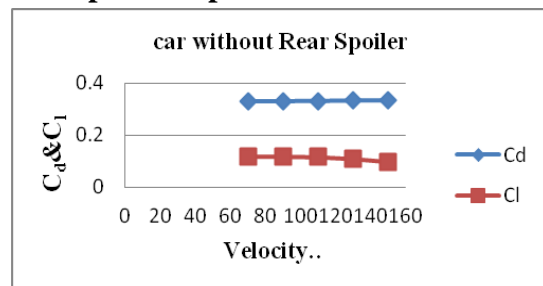


Figure 17: Variation of lift forces & drag forces with increase in speed for without spoiler

7.2. Variation of lift and drag forces with increase in speed for without Spoiler:

Figure 17 shows the Variation of lift forces & drag forces with increase in speed for without spoiler. It can be observed that the drag force increases and the lift force decreases slightly with increase in speed of the car.

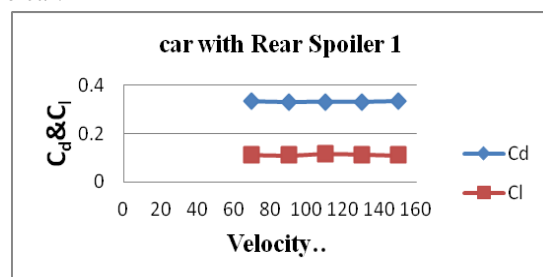


Figure 18: Variation of lift forces&drag forces with increase in speed for with spoiler1

7.3 Variation of lift forces & drag forces with increase in speed for with spoiler1:

Figure 18 shows the Variation of lift forces & drag forces with increase in speed for With spoiler1. It can be observed that slight fluctuation in drag force and lift force with increase in speed of the car.

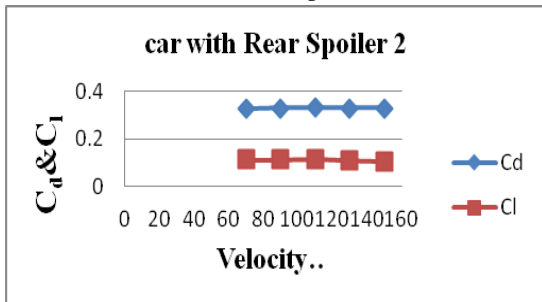


Figure 19: Variation of lift forces&drag forces with increase in speed for with spoiler2

7.4 Variation of lift forces & drag forces with increase in speed for with spoiler2:

Figure 19 shows the Variation of lift forces & drag forces with increase in speed for with spoiler2. It can be observed that slight fluctuation in drag force and lift force with increase in speed of the car.

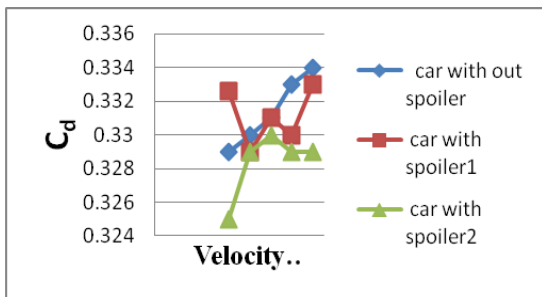


Figure 20: Comparison of drag forces for 3 cases with increase in speed

7.5 Comparison of drag forces for 3 cases with increase in speed:

- Figure 20 shows the Comparison of drag forces for 3 cases with increase in speed.
- From the above results the spoiler 2 yields the better results compared to spoiler1 and without spoiler because of diminishment of recirculation zones.
- The coefficient of drag for rear end of car body with spoiler 1 and without spoiler at 110 Km/hr yields the similar results and also the coefficient of drag for rear end of car body with spoiler 1 and ith spoiler 2 at 90 Km/hr yields the similar results because of constant stream lines

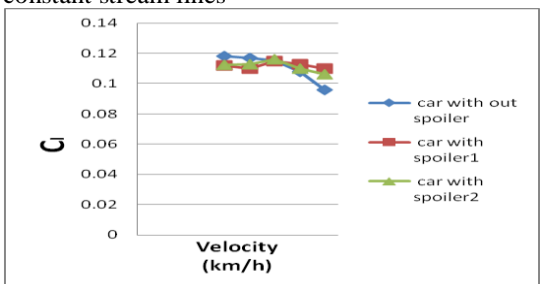


Figure 21: Comparison of lift forces for 3 cases with increase in speed

7.6 Comparison of lift forces for 3 cases with increase in speed

- Figure 21 shows the Comparison of lift forces for 3 cases with increase in speed.
- The coefficient of lift for body without spoiler is diminishing gradually.
- The C_l for body with spoiler 1 yields the increment compared with body with spoiler 2 is nearer to results with the both body without spoiler and with spoiler 1.
- The spoiler 2 is preferred because there is no rapid increment or decrement in C_l and reduction in coefficient of drag is also taken into consideration.

7.7. Reasons for selection of car body with spoiler 2

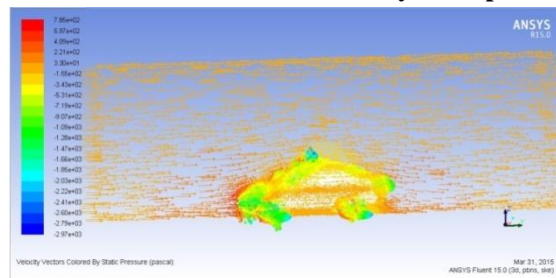


Figure 22: Velocity streamlines of flow in the symmetry plane for case 1

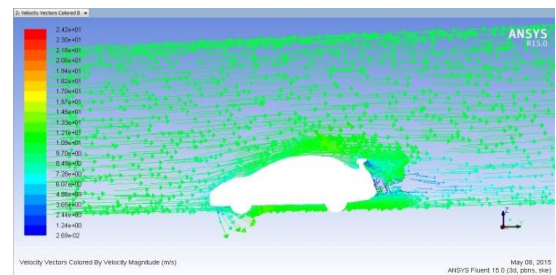


Figure 23: Velocity streamlines of flow in the symmetry plane for case 2

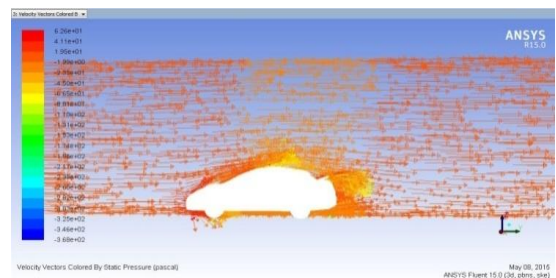


Figure 24: Velocity streamlines of flow in the symmetry plane for case 3

Contours of velocity for a high-speed vehicle at the symmetry plane for all 3 cases are shown in Figure 22, Figure 23 and Figure 24 respectively. The vehicles with rear spoiler (case 2 and case 3) have large and double air swirls at the rear end. It has been found that there were recirculation zones behind the rear end of the vehicle. By comparing the cases in figures, the recirculation zone behind the rear end of vehicle with spoiler situations (case 2 and case 3) were clearly larger. As we have seen in the Figure 5.8 there

were two different recirculation zones at the rear end of the vehicle (one behind the vehicle, and one above the rear window). By comparing Figure 22 and Figure 23 it has been seen that; the recirculation zone above the rear window was almost gone by using spoiler. From the figure 23 and figure 27 it is concluded that the recirculation zones in figure 27 have negligible impact on the performance because its magnitude is less.

8. Conclusions

Based on this study of the aerodynamic flow around a car, second spoiler of the car stood best among the three cases of car based on results obtained in terms of Drag it is about 0.329 and Lift force is 0.106. To estimate the drag coefficient and flow visualization is achieved successfully by the use of CFD simulations. Aerodynamics drag for car body of profiles are successfully simulated with ANSYS Fluent solver, the analysis shows aerodynamics drag in term of drag forces or drag coefficient proportionally increased to the square of velocity for car body. This computational analysis shows that there is possibility of improving the aerodynamic performance of car by modifications in exterior design of car body. These modifications are helpful in reducing the coefficient of drag i.e. Cd which effects the fuel consumption. By these modifications the coefficient of drag is reduced by approximately 2.18%

Advantages of using spoiler2:

- Increases tires capability to produce cornering force
- Stabilizes vehicles at high speed
- Improves braking performance
- Gives better traction

References

- [1] Wolf-Heinrich Hucho: Aerodynamic of Road Vehicle; Fourth Edition; Society of Automotive Engineers, Inc. 1998.
- [2] Heinz Heisler: Advanced Vehicle Technology: Second Edition; Elsevier Butterworth Heinemann. 2002.
- [3] Rosli Abu Bakar, Oleg Zikanov, Fazli Ismail, Design and Development of Hybrid Electric Vehicle Rear Diffuser, *Science, Technology & Social Sciences* 2008 (STSS), Malaysia.
- [4] Luca Iaccarino. Cranfield University Formula 1 Team: An Aerodynamics Study of the Cockpit. School of Engineering. Cranfield University. August 2003.
- [5] Carr. G. W., (1982), "The aerodynamics of basic shapes of road vehicles, part 1, Simple rectangular bodies", MIRA report No.1982/2.
- [6] Manan Desai1, et al/,"A Comparative Assessment of two Experimental Methods for Aerodynamic performance Evaluation of a Car" *Journal of Scientific & Industrial Research* 2008.
- [7] Aniket A Kulkarni et al Analysis of Flow over a Convertible Computer-Aided Design & Applications, *PACE* (2), 2012, 69-75,2012 CAD Solutions, LLC
- [8] Mohan Jagadeesh Kumar M et.al Effect of Vortex generators on Aerodynamics of a Car: CFD Analysis vol-2 issue 2 april-2013.ISSN-2319-1058
- [9] K. Lowe. Automotive steels. Engineering, Feb. 1995, 20-21.
- [10] Baker, C. The flow around high speed trains. *Journal of Wind Engineering and Industrial Aerodynamics* 2010; 98: 277-298.
- [11] Mahmoud Khaled, Hischam., Fabien, and Hassan P. Some innovative concepts for a car reduction: Parametric analysis of aerodynamic forces on simplified body.
- [12] Simon Watkins, Gioacchino Vio. The effect of the vehicle spacing on the aerodynamics of a representative vehicle. *Journal of Wind Engineering and Industrial Aerodynamics* 96 (2008).
- [13] Jasinski, W. and Selig, M., Experimental Study of Open - Wheel Race-Car Front Wings, *SAE Technical Paper* 983042, 1998, doi:10.4271/983042
- [14] Fundamentals of Fluid Mechanics by Munson, Young, Okiishi, Huebsch: 515,513.
- [15] Anderson CFD book: the basic with application McGraw-Hill series in aeronautical and aerospace engineering.
- [16] ANSYS® Academic Research, Release 13.0, Help System, FLUENT Theory guide, ANSYS, Inc.
- [17] Ahmed, G. Ramm, and G. Faltin. Some salient features of the time averaged ground vehicle wake. *SAE Paper* 1984; 840300.
- [18] Bayraktar, I., Landman, D., and Baysal, O. Experimental and Computational Investigation of Ahmed Body for Ground Vehicle Aerodynamics. *SAE Technical Paper* 2001; 01: 2742.
- [19] Baker, C. A., Grossman, B., Haftka, R. T., Mason, W. H., & Watson, L. T. HSCT configuration design space exploration using aerodynamic response surface approximations. *AIAA* 1999; 98: 4803.
- [20] Braus, M., Lanfrit, M. (2001). Simulation of the Ahmed Body. 9th ERCOFAC/IAHR Workshop on Refined Turbulence Modelling.
- [21] Guo, L., Zhang, Y., & Shen, W. Simulation Analysis of Aerodynamics Characteristics of Different Two-Dimensional Automobile Shapes.

Journal of Computers, 2011; 6(5), 999-1005.
doi:10.4304/jcp.6.5.999- 1005.

- [22] Peddiraju, P., Papadopoulous, A., Singh, R. (2009). CAE framework for aerodynamic design development of automotive vehicles. 3rd ANSA & μ ETA International Conference.
- [23] S. N. Singh, L. Rai, A. Bhatnagar. "Effect of moving surface on the aerodynamic drag of road vehicles" Proceeding of IMech E. J Automobile Engineering, 2004; 219 Part D: 127- 134.
- [24] Samareh, J. A. Aerodynamic shape optimization based on free-form deformation. *AIAA*, 2004; 4630.
- [25] Wolf-Heinrich Hucho (1998) "Aerodynamics of Road Vehicles", SAE International, Warrendale

Websites

- 1) www.wikipedia.com
- 2) www.carsdesignonline.com

Biographies



V. Naveen Kumar

Master of Engineering in Machine Design, Mechanical Engineering, Sir.C.R. Reddy College of Engineering, Andhra Pradesh, India



Dr.K. Lalit Narayan

Professor, Dept of Mechanical Engineering, Sir.C.R.Reddy College of Engineering, Andhra Pradesh, India.



L.N.V. Narasimha Rao

Assistant Professor, Dept of Mechanical Engineering, Sir.C.R.Reddy College of Engineering, Andhra Pradesh, India.



Y.Sri Ram

Assistant Professor, Dept of Mechanical Engineering, Sir.C.R. Reddy College of Engineering, Andhra Pradesh, India.