Aerodynamic Drag Reduction on Vehicle with and without Spoiler

Rajath. H.R[1] Mrs. Shweta Agrawal[2]
[1]P.G. Student [2]Assistant Professor
[1,2]Department of Mechanical Engineering
[1,2]MVJ College of Engineering Bengaluru

Abstract— Present days the demand for high speed passenger vehicles is increasing but doing so involves dealing with vehicle stability concerns. Forces like Drag, Lift and weight acting over a vehicle when moving on road have significant impact over the fuel consumption. So it is important to optimize these parameters to obtain greater efficiency. Spoiler is such a device which is used to increase fuel efficiency by reducing drag and also improving the down force. Adding a spoiler at the very rear of the vehicle makes the air slice longer, gentler slope from the roof to the spoiler, which helps to delay and reduce the flow separation. Reducing flow separation decreases drag, which increases fuel economy and less pollution as well; Also a spoiler is in a shape of inverted airfoil produces negative lift (down force) which helps to keep the vehicle attached to the ground at high speeds and corners. This thesis will present a simulation of flow around Passenger vehicle with spoiler positioned at the rear end using commercial fluid dynamic software. The thesis will focus on CFD-based lift and drag prediction of a passenger vehicle with and without spoiler, where the spoiler is mounted at the rear edge of the vehicle. A 3D computer model of 4-door passenger vehicle (which is designed with commercial CAD software Catia®) will be used as the base model. Different spoilers will be positioned at the rear end of vehicle to reduce the overall drag produced and the simulation will be performed using CFD code CFX to determine the aerodynamic effects of spoiler.

Key words: Passenger Vehicle, Rear Spoiler, CFD

I. Introduction

At present, altered vehicle turns out to be more main stream around the globe. Aerodynamic characteristics of altered vehicle are in this way definitely of critical interest. Well design of vehicle clearly gives a decrease in car accident and fuel utilization. The execution, handling, safety, and solace of a vehicle are essentially influenced by its aerodynamic properties. Additional parts are added to the body like rear spoilers, lower front and back bumpers, air dams and numerous more optimal design helps as to direct the air stream in various way and offer more prominent drag reduction to the vehicle and in the meantime improve the stability.

In case of that, numerous aerodynamics helps are sold in business sector for the most part rear spoiler. Rear spoiler is a segment to increase down force for vehicle particularly passenger vehicle. It is a aerodynamic device that design to spoil" unfavorable air development over an vehicle body". Fundamental altering area is at rear portion, relies on upon state of the rear portion either the vehicle is square back, notch back or fastback because not all rear spoiler can be fix at any type of rear portion of a car. However spoiler additionally can be appended to front back guard as air dam. Back spoiler contributed some real optimal design element which is lift and drag. The decrease of drag force can save fuel; additionally spoiler likewise can be utilized to control stability at cornering. Other than can reduce drag and decrease rear-axle lift, back spoiler additionally can diminish dirt on the rear surface.

In the process of vehicle design and alteration, the aerodynamics must be genuinely considered. A passenger vehicle design must be satisfactory if its structure drag reduced. Numerous specialists have made utilization of CFD procedures to perform investigation on changed passenger vehicle to improve a superior result. The present study introduces the improvement procedure of aerodynamic holography in the vehicle external body. A few CFD examinations were performed to analyze the pressure field, velocity vector field, and aerodynamic force expectation identified with a passenger vehicle. Through Ansys CFX, the flow properties around the vehicle can be computed. This paper shows an Analysis of flow around modified vehicle with spoiler situated at the rear side utilizing commercial fluid dynamics software Ansys CFX. The study concentrates on CFD-based lift and drag expectation with respect to the vehicle body and a improvement in the design because of spoiler designs.

II. Methodology

In this work, initial a model of the Passenger vehicle is set up in the CATIA programming and this generic model is import into the ANSYS CFX to do the simulation of the coefficient of drag and coefficient of lift in the wind tunnel which is made in the outline module of the ANSYS CFX. After this the meshing is produced on the surface of the passenger vehicle. Aerodynamic evaluation of air flow over an object can be performed utilizing Computational Fluid Dynamics (CFD) approach.

III. Model Setup

The base line model of generic passenger car is designed in CATIA software. Figure 1 show the generic passenger car used in the CFD simulation. The full size generic passenger car is 4652mm long, 1800mm wide, 1449mm high. Then after, this model has been analyzed to determine the properties of flow around the passenger car under ANSYS CFX.

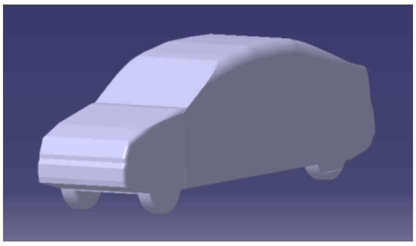

Fig. 1: CATIA model of a car without spoiler

A. Spoiler Dimensions

Two different kinds of spoiler were used. First one is " wing" type spoiler, fitted 23 cm above from the surface of rear end of vehicle, Second spoiler fitted border of ending side of automobile without any space between exterior of automobile and spoiler. Figure 2 shows spoiler 1 and figure 3 shows spoiler2 with dimensions.

1) *Spoiler 1: CATIA model of spoiler 1*

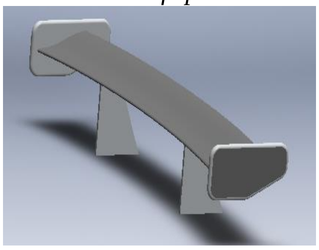

Fig. 2: Catia model of Spoiler 1

2) *Spoiler 2: CATIA model of spoiler2*

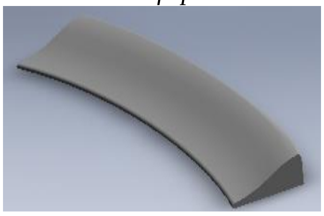

Fig. 3: CATIA model of spoiler 2

With the help of "assembly" functionality spoilers has been fitted at the rear end of the passenger car in Catia. These automobile models are used for Analysis.

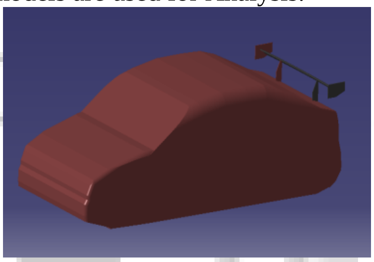

Fig. 4: Assembled 3D Catia model of spoiler 1 vehicle

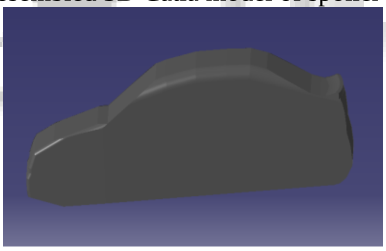

Fig. 5: Assembled 3D Catia model of spoiler 2 vehicle

Figure 1, Figure 4 and Figure 5 shows 3-D computer models of Passenger car and Car with spoilers respectively. A virtual air-box has been created around the 3D CAD model (Figure 6), 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.

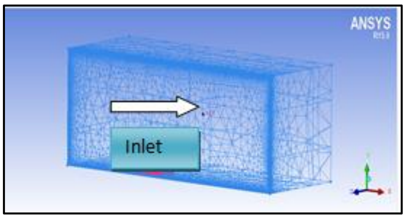

Fig. 6: Virtual wind tunnel

Because of the complexity of the simulation with limited PC assets and time, the complete domain was divided to half utilizing a symmetry plane (YZ plane), which implies, the simulation would be computed for simply the one side of the vehicle and since the other side is symmetric and YZ plane has been characterized as symmetric boundary in the solver to make the boundary condition as "a slip wall with no shear forces", the simulation results would be valid for full model too. All surfaces of the virtual wind tunnel (air-box) have been named so the numerical solver of ANSYS CFX would remember them and apply the proper boundary conditions automatically. The final meshing can be seen in Figure 7. The same procedure has been used to create high dense meshing followed for all cases. Case 1: Vehicle without spoiler, Case 2: Vehicle with first spoiler, Case 3: Vehicle with second spoiler, exactly the same.

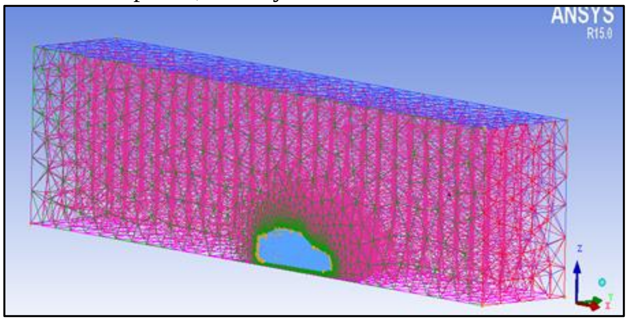

Fig. 7: Final mesh of a virtual wind tunnel

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

Table 1: Mesh sizing parameters

IV. CFD ANALYSIS

Computational Fluid Dynamics can be defined as" Obtaining an approximate solution to the problem of Heat transfer and fluid dynamics by using a set numerical procedure". As this definition states it is a branch of science by itself, It is a path of integrating numerical methods one (numerical analysis) after other (Heat and mass transfer). In respect computational fluid dynamics is an integration of Fluid mechanics, mathematics and a computer science. In order to obtain a solution Researchers Converting to a Software language or computer programs from a high-level languages to solve mathematical equations". The word computational means the investigation of fluid movement through numerical recreations, which include software package or computer programs to attain the required result.

CFD provides an alternative and experimental approach by giving cost effective simulation result on fluid flow. Particularly CFD reduces time and designing costs and production when compare to experimental method. It offers an ability of solving a tough problem over fluid flow in a mean time compare to analytical method. A detailed, comprehensive information and visualized results can be obtained in CFD compare to experimental and analytical method of fluid dynamics.

A. Solver Settings for CFD Simulation

Cases	Case 1	Case 2	Case 3
Simulation			
Fluid properties	Type of fluid Air		
	Density $\rho = 1.175\ kg\ m$		
	Kinematic viscosity $\nu = 1.8247x10\ kgm{\cdot}s$		
Domain motion	Stationary		
Reference pressure	1.0000e+00atm		
Heat transfer model	Total energy		
Turbulence model	k-Epsilon		
Turbulent wall functions	Scalable		

Table 2: Solver settings of Passenger vehicle

Boundary condition		
Velocity Inlet	Flow Regime	Subsonic
	Static temperature	2.8815e+01 m/s
	Normal speed	2.7778e+01 m/s
	Turbulence	Medium intensity and Eddie viscosity ratio
Outlet	Relative pressure	1.0000e+00 Pa
Wall zones	No slip	
Symmetry	No slip	

Table 3: Solver settings of Passenger vehicle

V. RESULT AND DISCUSSION

A stable condition, incompressible simulation explanation of N-S equations was simulated by with ANSYS-CFX. k-ε model has been used for turbulence modeling using wall functions of scalable. The results obtained for all the cases meshing resolution is same, boundary condition and the same turbulence modeling. The velocity has been set up in boundary condition is 200km/hr is same for all cases.

A. Case1: Passenger Vehicle without Spoiler

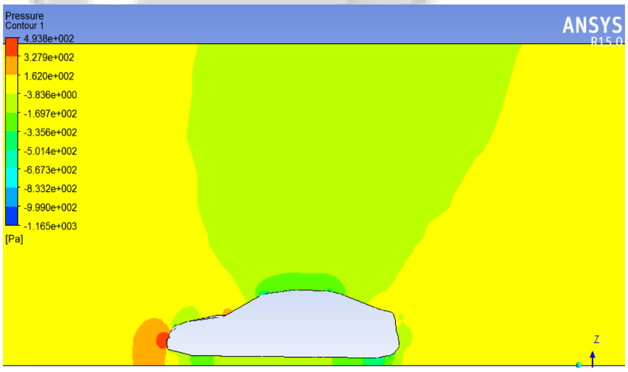

Fig. 8: Pressure contour of Passenger vehicle

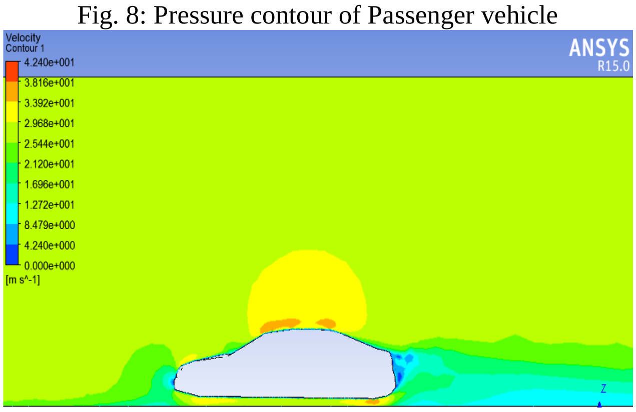

Fig. 9: Velocity contour of Passenger vehicle

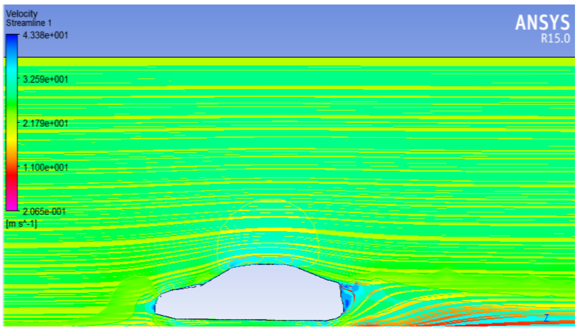

Fig. 10: Streamline contour of Passenger vehicle

1) Drag and Downforce

Drag is a force acting opposite to the relative development of any article moving with respect to an enveloping liquid. This can exist between two fluid layers or a fluid and a solid surface. The resultant Drag force obtained is

Co-efficient of drag, Cd = 0.2911

Downforce is a downwards pushed made by the streamlined traits of an auto. The inspiration driving downforce is to allow an vehicle to travel speedier through a corner by growing the vertical force on the tires, in this manner making more handle.

The resultant Downforce obtained for this Passenger vehicle is

Cl = -0.5544

B. Case 2: Passenger Car with Spoiler1

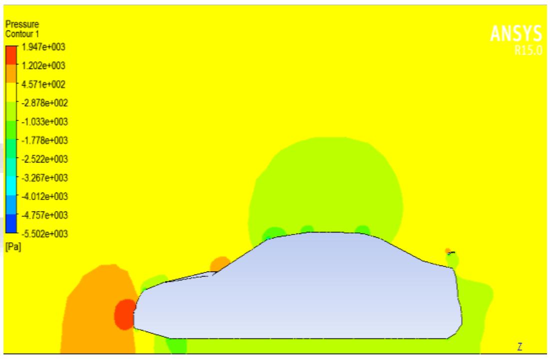

Fig. 11: Pressure contour of Passenger vehicle with spoiler 1

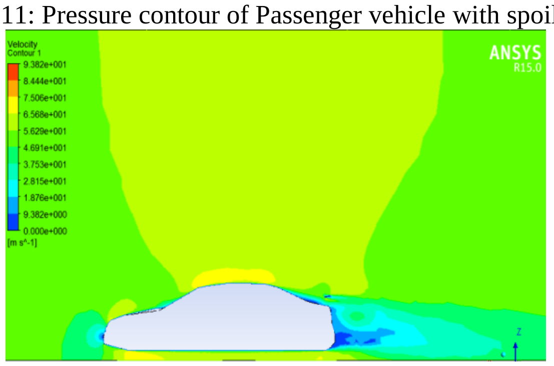

Fig. 12: Velocity contour of Passenger vehicle with spoiler 1

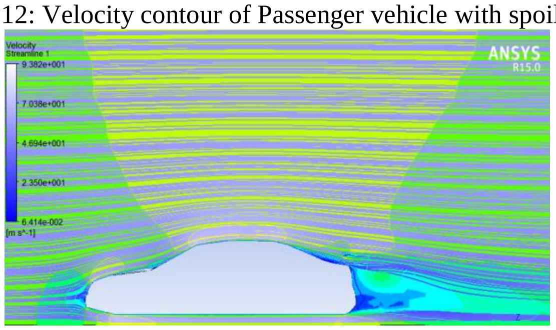

Fig. 13: Streamlines of Passenger vehicle with spoiler 1

1) Downforce and Drag:

The resultant Downforce obtained for this Passenger vehicle with spoiler 1 is:

Cl = -0.101

The resultant Drag force obtained is :

Cd = 0.2779

Even though this spoiler has reduced the overall downforce of the car when compared to without spoiler case it can also decreased the overall drag coefficient.

C. Case 3: Passenger car with spoiler 2

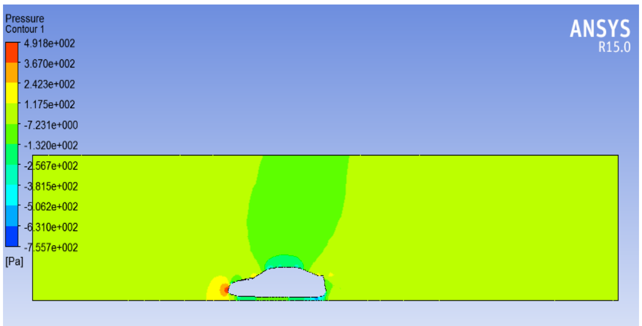

Fig. 14: Pressure contour of Passenger vehicle with spoiler 2

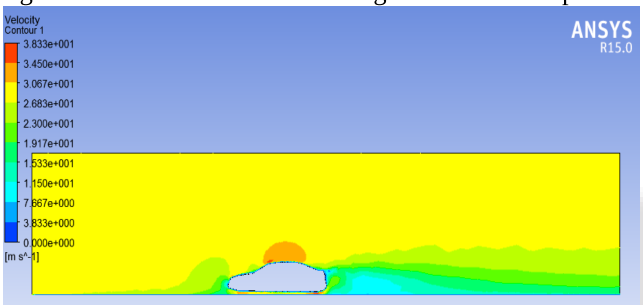

Fig. 15: Velocity contour of Passenger vehicle with spoiler 2

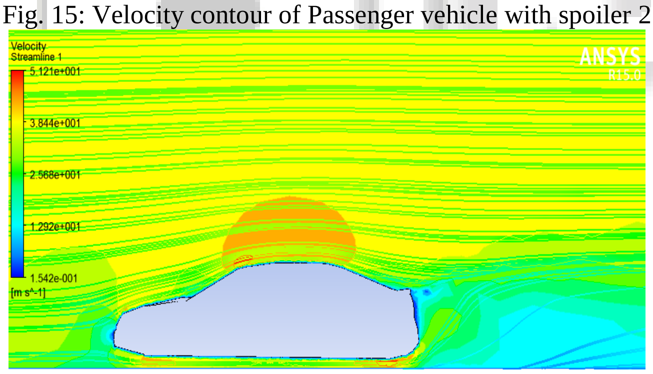

Fig. 16: Streamline of Passenger vehicle with spoiler 2

1) Downforce and Drag

The resultant Downforce obtained for this Passenger vehicle spoiler 2 is:

Cl = -0.225

The resultant Drag force obtained is

Cd = 0.278

The down force is lesser compared to the car without spoiler but more when compared with the spoiler 1. Overall Drag co-efficient is lease when compared to Passenger vehicle without spoiler and more compare to vehicle spoiler 1. The overall Cl has marginally reduced but the Cd has also reduce due to the introduction of spoiler.

VI. Conclusion

Computational Fluid dynamics simulation using ANSYS CFX to predict flow around modified passenger car has been achieved. Three cases were simulated with 3 different spoilers for a passenger vehicle. Through comparing the results of these cases, a rear spoiler of better performance has been developed. By considering passenger vehicle with spoilers 1 and 2 and without spoiler:

The numerical analyze of passenger vehicle with spoiler 1 and spoiler 2 has showed that, the aerodynamic drag of passenger vehicle with spoiler 1 is condensed from 0.2911 to 0.2779, which is 4.5% drag decrease, and co-efficient lift has been reduced to -0.5544 to -0.101. The aerodynamic drag of passenger vehicle with spoiler 2 is condensed from 0.2911 to 0.278, which is 4.49% drag decrease, and co-efficient of lift has been reduced to -0.5544 to -0.225.

By looking at figure 10, 13 and 16 it has been found that, the distribution region over the back glass was verging on passed by utilizing spoiler 2. The fluid slanted tenderly over the back glass, which helps maintaining the back window more clean.

In these results with spoilers 1 and 2 of passenger vehicle shows a drag reduction of a vehicle, which would result in fuel saving. Spoiler 1 is effective and reduces the Drag force for passenger vehicle, Spoiler 2 also decreases the drag force for different slant angles when compared to without spoiler but it produces better down force among the other spoiler.

VII. Future Work

This Simulation can be carried out with different spoilers to obtain a better result in Drag force and Downforce, and also can be tested for different spoiler position and Different rear slant angle in a vehicle and for different speeds. Simulation can be tested to any kind of vehicle or turbulence model vehicle and with refined mesh which gives a good result. And experimental analysis can be carried out with different speeds.

Reference

[1] John J Bertin, "Aerodynamics for Engineers", prentice hall; 5th edition, New jersey, june 2008.

[2] Jiyuan Tu, Guan Heng Yeoh and Chaoqun Liu, "Computational Fluid Dynamics: A practical Approach", Butterworth-Heinemann; 1st edition, Burlington, MA, Nov-2007.

[3] Oleg Zikanov, "Essential Computational Fluid Dynamics", John wiley and sons, lnc. Hoboken, New jersey, March 2010.

[4] C H K Williamson, "Three dimensional vortex Dynamics in bluff body wakes", Experimental thermal and fluid science, volume 12, Feb 1996, p.150-168.

[5] Wolf-Heinrich Hucho, " Aerodynamics of road vehicles: from Fluid Mechanics to vehicle engineering", Society of Automotive engineers Inc; 4th edition, Warrendale, pa, Feb-1998.

[6] Gilhaus, R. Hoffmann, Directional stability, Aerodynamics of Road Vehicles, in: W H Huccho(ED), SAE international, Warrendale, 1998.

[7] J R Callister, A R Geiarge, wind noise, Aerodynamics of road vehicles, in: W H Hucho(ED), SAE international, warrendale, 1998.

[8] John D Anderosn JR "Fundamentals of Aerodynamics", Tata Mcgraw hill publishing company pvt ltd, New delhi, 2007.
[9] John D Anderson "Fundamentals of Aerodynamics", Tata Mcgraw hill publishing company pvt ltd, New Delhi, 2007
[10] Hucho WH. "Aerodynamics of road vehicles" SAE international, 1998.
[11] Dan Barbut, Eugen Mihai Nergus, CFD analysis for road vehicles- care study, Incas Bulletin, 3(2011) 15-22.
[12] M Roumeas, P Gilleron and A kourta, Drag reduction by separation control on a car after body, International journal for numerical methods in Fluids, 60(2009) 1222-1240.