CFD Analysis of Rear Spoiler of Passenger Car

Minal Pandurang Marathe[1] Anil Elisala[2]
[1]Research Scholar [2]Professor
[1,2]Department of Mechanical Engineering
[1,2]B.M. College of Engineering, Indore

Abstract— With increase in the number of accidents, the automotive industry always strive for measures to improve safety of passenger vehicles at high speeds. Proper orientation of spoiler gives maximum down-force with least drag-force .The aerodynamic performance of Formula-One car is determined by measuring the drag-force and down-force acting on the car. Various geometric models of the car are created in the CAD package CREO 2. The CFD analysis is done by using ANSYS CFX. It shows the flow pattern over the car. The behavior of the flow is greatly influenced by the spoilers which were simulated with the rear end of car and therefore the design has to take into account the environment of the spoilers. The spoilers demonstrate their utilities as deflectors that are able to deflect the flow downward.

Key words: Disc Brake, CFD, Transient thermal analysis, Ansys CFX

I. INTRODUCTION

While designing a passenger cars the most important and significant aspects the designer has to deal with, is the field of aerodynamics. The aerodynamic designer has two primary concerns: the creation of down force, to help the car steer onto the track and improve cornering forces; and minimizing the drag force, caused by turbulence which in turn decreases the speed of the car .These factors enhance the performance of the car. The rear spoiler of the passenger car is one of the crucial aerodynamic component. Down force is generated by the spoilers due to the ground effect where more force is generated when aerofoil is moving close to the ground surface. The spoiler design is a very crucial in which small improvements in the design of the car may largely affect the car's performance. Hence the use of robust and concrete technology is necessary to produce the best possible design for higher performance.

Fig. 1: Spoiler

II. LITERATURE SURVEY

A lot of research is being carried out from the last decade in automotive field and the safety is one of the main concerns and is always play an important role in a vehicle anatomy. Spoilers are aimed to increase downward drag to ground to get better traction and avoid skidding of vehicles at high speeds.

Chien-Hsiung Tsai a, Lung-Ming Fu b, Chang-Hsien Tai a, Yen-Loung Huang a, Jik-Chang Leong, [2007] [1] The author in reference [1] has proposed an effective numerical model based on the Computational Fluid Dynamics (CFD) approach to obtain the flow structure around a passenger car with wing type rear spoiler. The results were obtained by comparing both cases i.e. car with and without spoiler. The angle of attack is different for different type of spoiler. Simulations are performed after adding different types of spoilers with different angles of installation.

Emmanuel Guilmineau, [2007] [2] The author in reference [2] has proposed the study based on Ahmed body. The Ahmed body is a simplified car used in automotive industry to investigate the influence of the flow structure on the drag. The model of a hatch back body is simulated using all k-ε models in CFD. Rear slant angle is taken into account. This paper investigates numerically the flow around the Ahmed body for the base slant angles 25 and 35. It is found that there is a 3% decrement in drag when different slant angles were used. Simulations, with several turbulence models, have been carried out for the generic Ahmed body with 25and 35 slant angles.

W. Kieffera, S. Moujaesb, N. Armbyab, [2007] [3] The author in reference [3] has proposed brief literature study on using the Star-CD CFD code to perform a turbulent simulation (using a k–ε model) of the airflow on the front and rear wings of a Formula Mazda car with different angles of attack and the effect of the ground on the front wing. Results showing pressure and velocity distributions and lift (Cl) and drag coefficients (Cd) for the different cases are shown graphically and the numerical model was set up. With the front aerofoil close to the ground and with 0 degree AOA, the velocity on the upper surface is slower than for the aerofoil in free air. On the lower surface of the wing, the velocity is higher for the aerofoil in ground effect than the one in free air. For 16 AOA, separation occurs at half of the chord length. The greater portion of lower surface is close to ground will creates larger low pressure area and hence greater is the down force.

Mohd Akmal, [2007] [4] The author in reference [4] has proposed the effect of aerodynamics on a hatchback car. This research is focused on two important aspects namely model development and wind tunnel experiment and the results of this research focused on qualitative and quantitative values. Quantitative values are Co-efficient of drag and pressure distribution around the model surface. While qualitative values are visualization of air flow characteristics over the model surface.

Hugo G. Castro, Rodrigo R. Paz, Mario A. Storti, Victorio E. Sonzogni, Jorge O, [2008] [5] The author in reference [5] has proposed experimental and numerical studies of the aerodynamic behaviour of simplified road vehicle, Experimental and numerical analysis of the flow over the Ahmed body with a 35 degree rear slanted angle were performed. Ahmed body was chosen as a standard reference given that it was numerically and experimentally studied in an exhaustive way. Comparing both, results from the experimental tests made in this work and the other experimental reference as well as the provided by the computational simulation presents some appreciable differences. Mean drag force in the experimental tests (Cd mean = 0:40) presents a high value when comparing with the obtained by Ahmed et al. (1984)(Cd mean = 0:257) while the numerical obtained value had a difference lower than a 9% (Cd mean = 0:28) been corrected for tare drag of stilts

III. PROBLEM FORMULATION

Aerodynamic evaluation of air flow over an object can be performed using analytical method or CFD approach. On one hand, analytical method of solving air flow over an object can be done only for simple flows over simple geometries like laminar flow over a flat plate. If air flow gets complex as in flows over a bluff body, the flow becomes turbulent and it is impossible to solve NavierStokes and continuity equations analytically. On the other hand, obtaining direct numerical solution of Navier-stoke equation is not yet possible even with modern day computers. In order to come up with reasonable solution, a time averaged Navier-Stokes equation is being used (Reynolds Averaged Navier-Stokes Equations – RANS equations) together with turbulent models to resolve the issue involving Reynolds Stress resulting from the time averaging process. In present work the k-e turbulence model with non-equilibrium wall function is selected to analyze the flow over the generic passenger car model. This k-e turbulence model is very robust, having reasonable computational turnaround time, and widely used by the auto industry.

A. *Modeling and CFD Analysis*

1) *Cad Modeling*

Initially a CAD model of passenger car is modelled in Creo. Creo is 3d modelling software developed by PTC

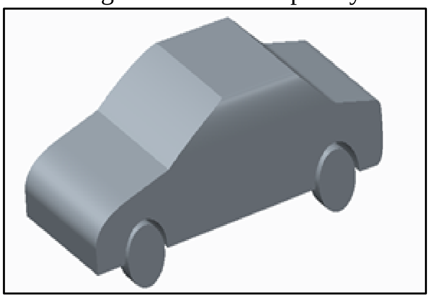

Fig. 2: Passenger car Model (no spoiler)

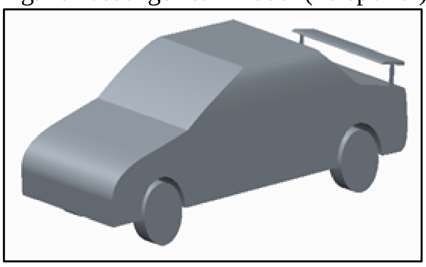

Fig. 3: Passenger car Model (spoiler)

2) *Computational Domain*

The CAD model in .igs format is imported in ANSYS CFX module where a continuum is defined and meshed using tetra elements.

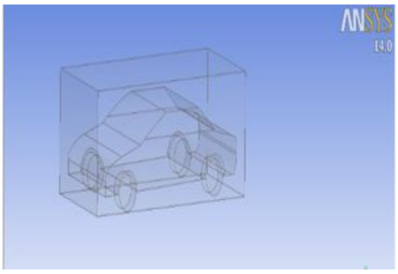

Fig. 4: Computational Domain

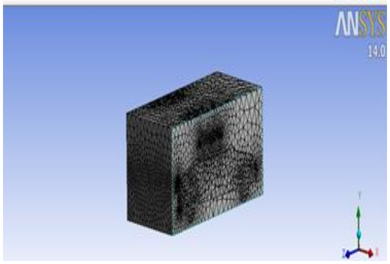

Fig. 5: Meshed model

Domain	Nodes	Elements
AIR1	33890	172843

Table 1:

3) *Boundary Conditions*

Domain is defined, inlet , outlet and opening boundaries are defined

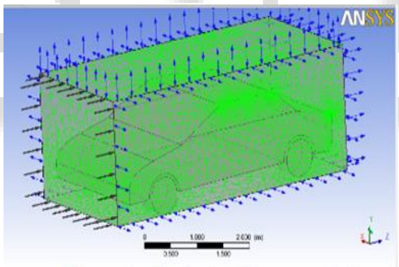

Fig. 6: Boundary Conditions

Domain - AIR1	
Type	Fluid
Location	Assembly
Materials	
Air at 25 C	
Fluid Definition	Material Library
Morphology	Continuous Fluid
Settings	
Buoyancy Model	Non Buoyant
Domain Motion	Stationary
Reference Pressure	1.0000e+00 [atm]
Heat Transfer Model	Isothermal
Fluid Temperature	2.5000e+01 [C]
Turbulence Model	k epsilon
Turbulent Wall Functions	Scalable

Table 2:

4) Solution

Setting up of convergence criteria r.m.s residual values are set to 1e-5 for pressure, velocity and energy

5) Postprocessing

Contour plots of pressure, velocity is plotted along with vector plot also to understand the flow across car and spoilers

IV. RESULT AND DISCUSSION

Pressure, lift force, drag force of entire vehicle is calculated by software at various speeds for both the cases with and without spoilers.

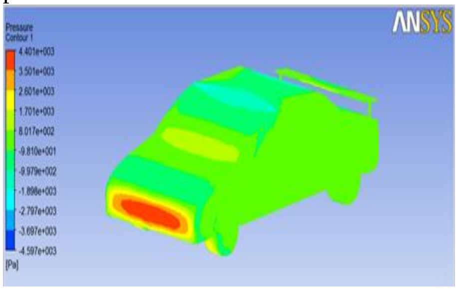

Fig. 7: Pressure distribution

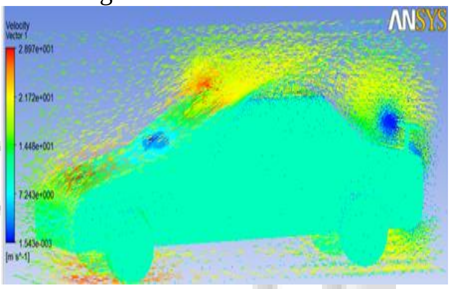

Fig. 8: Vector plot of velocity

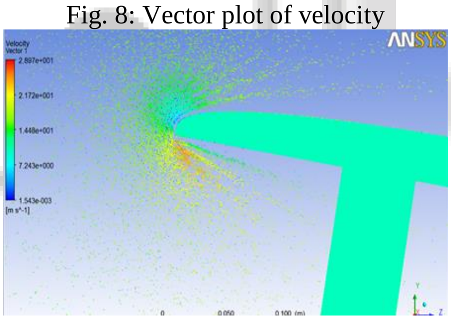

Fig. 9: Vector plot over and underneath spoilers

A. *CASE 1: Without Spoilers*

AIR VELOCITY (m/sec)	PRESSURE (Pa)	DRAG FORCE (N)	LIFT FORCE(N)
15.5	513.41	460.887	-33.89
25.5	1551	1241.5	-88.37
35.5	2691	2399.47	-167.469
45.5	4419	3934.14	-270.821

Table 3:

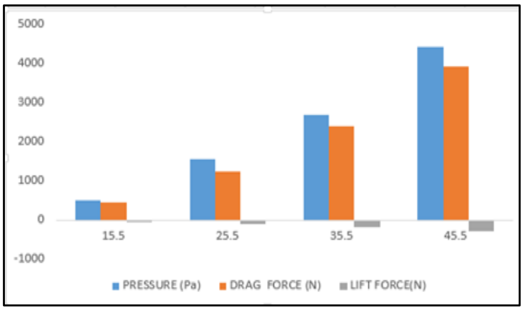

Fig. 10: Bar graph of pressure , velocity

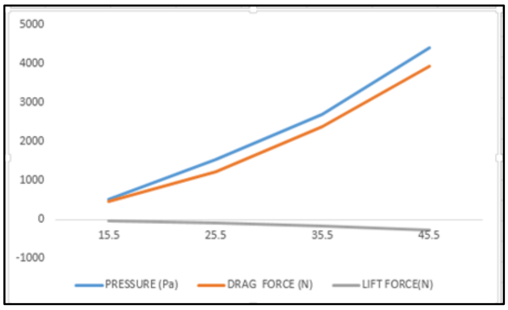

Fig. 11: Bar graph of pressure, velocity

B. *CASE 2: With Spoilers*

Air velocity(m/sec)	Pressure (Pa)	Drag Force	Lift Force
15.5	511.3	460.99	-58.17
25.5	1383	1241.61	-154.79
35.5	2680	2399.49	-279.01
45.5	4401	3933.95	-484.94

Table 4:

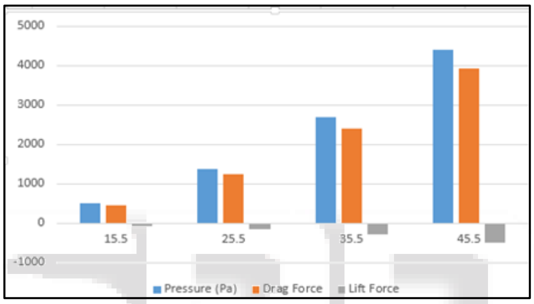

Fig. 12: Bar graph of pressure, velocity

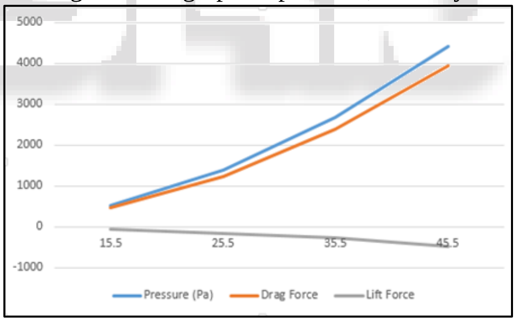

Fig. 13: Bar graph of pressure, velocity

V. CONCLUSION

1) CFD analysis of passenger car is performed and pressure, velocity, lift force, drag force experienced by vehicle is determined by ANSYS software.
2) With increase in air velocity the lift force, drag force and pressure experienced by car increases.
3) Without using spoilers all the parameters showed lesser value when compared with car design with spoilers.
4) With use of spoilers 75% increment in negative lift is observed.
5) Spoilers improved traction and plays a crucial role in avoiding skidding of vehicles at high speeds.

VI. FUTURE SCOPE

1) Further scope lies in improving design of spoilers to increase negative lift.

2) Location of spoiler could also play a major role in generating negative lift and could be analyzed.

REFERENCES

[1] Chien-Hsiung Tsai a, Lung-Ming Fu b, Chang-Hsien Tai a, Yen-Loung Huang a, Jik-Chang Leong "Computational aero-acoustic analysis of a passenger car with a rear spoiler" in [2007].
[2] Emmanuel Guilmineau "Computational study of flow around a simplified car body" in [2007].
[3] W. Kieffera, S. Moujaesb, N. Armbyab "CFD study of section characteristics of Formula Mazda race car wings" in [2007].
[4] Mohd Akmal, Hanif Bin Chik "Experimental study of aerodynamics of a hatchback car" in [2007].
[5] Hugo G. Castro, Rodrigo R. Paz, Mario A. Storti, Victorio E. Sonzogni, Jorge O "Experimental and numerical studies of the aerodynamic behavior of simplified road vehicle" in [2008].

