A Review on Design Optimization of Air Dam for Lift Reduct on of a Car using in Wind Tunnel and CFD Analysis

Sanat M. Gameti[1] Prof. H.P. Patel[2] Mayank K. Paramar[3]
[1,3]P.G. Student [2]Associate Professor
[1,2,3]Department of Mechanical Engineering
[1,2,3]L D College of Engineering, Ahmedabad

Abstract— Aerodynamic force plays an important role in vehicle performance and its stability when vehicle reaches higher speed. Nowadays the maximum speed of car has been increased above 180 km/hr but at this speed the car has been greatly influenced by drag and lift forces. So the researchers are mainly focused in reduction of co-efficient of drag and lift in car model at higher speed. Even though the various techniques are found by researchers for improving vehicle performance and its stability still we are in need of further improvement. So we are implementing “AIR-DAM” or “FRONT SPOILER” as an aerodynamic add on device at front portion of vehicle. The various shape of air dam are analyzed to obtain the efficient one to reduce the aerodynamic lift forces. MIRA car reference fast back model is used to analyses to Air-dam.

Key words: Aerodynamic, Air Dam, Wind Tunnel, CFD

I. Introduction

A. *Aerodynamics*

Aerodynamics is a branch of Fluid dynamics concerned with studying the motion of air, particularly when it interacts with a solid object, such as an airplane wing. Aerodynamics is a sub-field of fluid dynamics and gas dynamics, and many aspects of aerodynamics theory are common to these fields. Automotive aerodynamics is the study of the aerodynamics of road vehicles. Its main goals are reducing drag and wind noise, minimizing noise emission, and preventing undesired lift forces and other causes of aerodynamic instability at high speeds. For some classes of racing vehicles, it may also be important to produce down-force to improve traction and thus cornering abilities. Car aerodynamic makes the car safer and make it more fuel efficient. It is use in most of the car today specially in race car because there is more need of aerodynamics in high speed car. Understanding the motion of air around an object (often called a flow field) enables the calculation of forces and moments acting on the object. In many aerodynamics problems, the forces of interest are the fundamental forces of flight: lift, drag, thrust, and weight. Of these, lift and drag are aerodynamic forces, i.e. forces due to air flow over a solid body. Calculation of these quantities is often founded upon the assumption that the flow field behaves as a continuum.

II. Wind tunnel

In this case in order to save time and money, test are performed on a geometrically scaled model rather than the full scale prototype. Thus additional care must be taken to a powerful tool to achieve the above. The complete similarity between the model and prototype can be attained by achieving the three similarities namly Geometric, kinematic, Dynamic similarity.

Fig. 1: Wind Tunnel

The pressure distribution over the car surface can be measured at various taping point generated on the model. The values can be noted directly from the manometer readings. The pressure taping arrangement in the wind tunnel is shown below.

It is a suction type wind tunnel.

The maximum rpm attained by the drive is 2500.

Rating of motor is 5 KW

Variable speed is 5-40 m/s

Blockage Ratio

The blockage ratio of model is the ratio of the frontal-area of the model to test-section-area. The blockage ratio should not exceed 15%

Frontal surface area of the model =10.84x9.48 =102.763 cm²

Blockage ratio = 102.763/900=0.1141

Formula Used

For velocity

$$V = \sqrt{\frac{2\, g\, hw}{hair}}$$

Finding co-efficient of drag and lift

$$F = C_d \frac{1}{2} \rho A V^2$$

$$F = C_L \frac{1}{2} \rho A V^2$$

Reynold number

$$Re = \frac{\rho\, v\, L}{\mu}$$

A. *The Reading From Wind Tunnel Experiment Are Tabulated Below.*

SR NO.	LIFT FORCE	DRAG FORCE	STATIC HEAD DIFFERENCE	V	C_d	C_L
1	0.299	0.4850	1.6	16	0.3012	0.186
2	0.385	0.6307	2.0	18	0.3093	0.189
3	0.493	0.8001	2.6	20	0.3180	0.196
4	0.602	0.9858	3	22	0.3238	0.198

5	0.739	1.1797	3.5	24	0.3256	0.204
6	0.901	1.4065	4.5	26	0.3308	0.212
7	1.082	1.6496	5.0	28	0.3345	0.220
8	1.265	1.9022	5.6	30	0.3360	0.224
9	1.494	2.1855	6.3	32	0.3393	0.232
10	1.716	2.5087	7.3	34	0.3450	.0236

Table 1:

III. COMPUTATIONAL FLUID DYNAMICS (CFD)

In order to start the analysis procedure, the computational domain should be created around the car mode las like a wind tunnel arrangement. In order to achieve the vest practices of CFD the wind tunnel domain is created with the specification of the measurements off the wind tunnel are 1000*300*300 in mm.

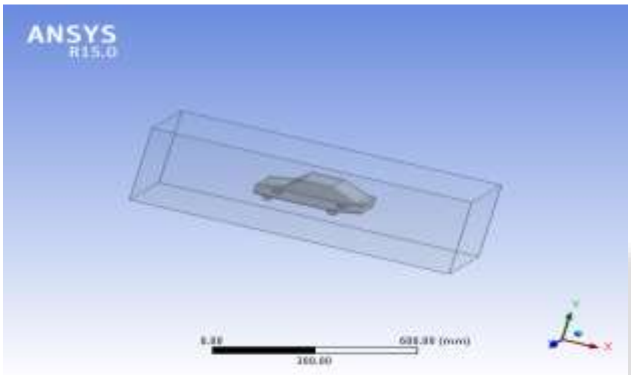

Fig. 2:

A. Volume Mesh

In the present numerical study, the vehicle component volume was discretized with tetrahedral mesh elements. A typical mesh element has size of 15-20 mm, and the smallest surface element has a size around 10mm. the mesh was generated with tetrahedral mesh elements with mesh elements 401426.

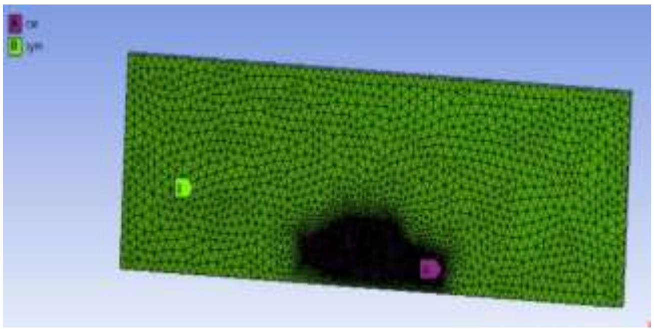

Fig. 3:

B. Boundary Condition

The various boundary condition is given in table, inlet condition, the velocity of incoming flow is given as V = 30 m/s, outlet condition, the gradient along main flowing direction is zero, the standard atmospheric pressure is put on exit boundary face. Blue and red faces indicate velocity inlet and pressure outlet respectively. White represents wall whereas yellow represents symmetry conditions.

Section	Physical condition	values
Inlet	Velocity	30 m/s
Outlet	Pressure	0-gauge pressure
Road (bottom)	Wall	-
Top	Symmetry	-

Table 2: Boundary Condition

Material
Fluid = Air
Density = 1.225 Kg/m^3
Interpolation method
Solver = fluent solver
Turbulence model = k - € turbulence model

IV. CFD RESULT

A. Co-Efficient Of Drag and Lift

The computational domain is extracted in the Fluent solver to calculate the co-efficient of drag under the applied boundary condition. The analysis result shows the CD value as 0.3301 and CL value as 0.202 with inlet velocity 30 m/s which matches well with the experimental result.

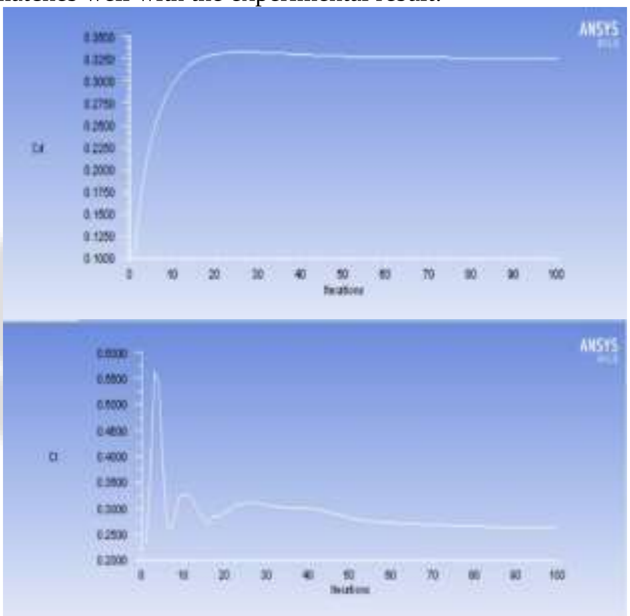

Fig. 4:

B. Static Pressure

Pressure contour demonstrating high pressure zones in the front of radiator portion and low in the regions with streamlined profile such as nose, vase of windshield etc. this pressure difference leads to pressure drop over the entire region

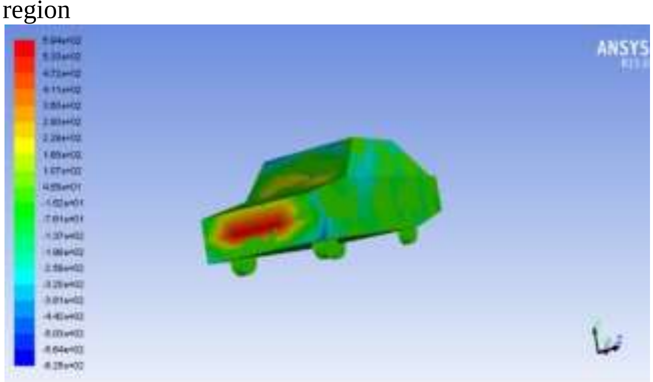

Fig. 5: Static Pressure

C. Contour of Velocity Vector

Above fig. shows the vector of air on a car body, the back of the car is in recirculation which is due to flow separation the happens due to uneven surface. The flow separation leads to air circulation.

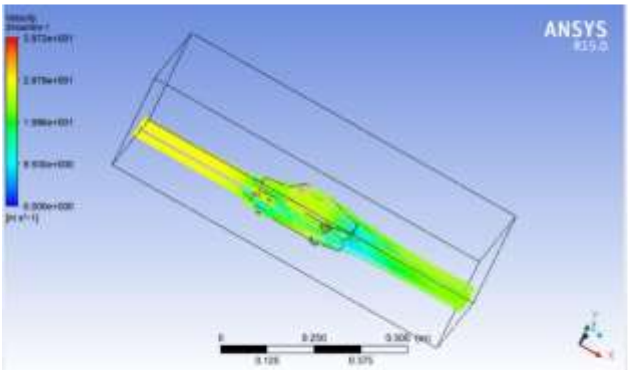

Fig. 6: Contour of Velocity Vector

V. AIR DAM

An air dam is usually a shaped part of the lower portion of the front bumper. It is sometimes called a "front lip" or "lip spoiler" or "front spoiler". It can be molded as part of the front bumper, or it can be a separate part that is attached at the front lower portion of the front bumper. Spoilers on the front of a vehicle are often called air dams, because in addition to directing air flow they also reduce the amount of air flowing underneath the vehicle which generally reduces aerodynamic lift and drag.

All AIR-DAM with the exception of in fig. were fitted to under the front bumper. The value of h, which was 136 mm, was also kept constant over the full width of the AIR-DAM. AIR-DAM A1, A2and A3 were respectively 45, 60 and 90 to the horizontal and were mounted immediately below the front portion.

After taking reading from wind tunnel results are tabulated below.

V	45		60		90	
	C_D	C_L	C_D	C_L	C_D	C_L
24	0.3138	0.166	0.3201	0.166	0.3238	0.166
26	0.3246	0.170	0.3246	0.175	0.3260	0.173
28	0.3279	0.182	0.3389	0.182	0.3396	0.183
30	0.3301	0.188	0.3325	0.190	0.3358	0.190
32	0.3386	0.201	0.3392	0.201	0.3400	0.201
34	0.3412	0.212	0.3423	0.213	0.3448	0.211

Table 3:

VI. CONCLUSION

From the above experimental result table shown that drag coefficient is reduced by some amount when we used the AIR-DAM. Most suitable AIR-DAM is at an angle of 45. The side AIR-DAM will restrict the air flow below the car and it will reduce the air flow and pressure and the coefficient of drag and lift. The drag coefficient is being reduced by 3% to 8% and lift co-efficient is reduce by 2% to 5%.

REFERENCES

[1] Aerodynamics improvement to the body and cooling system of a typical small saloon car, W.R. Stapleford journal of wind engineering and industrial aerodynamic, 9 (1981) 63-75
[2] Enhancement of aerodynamic characteristic in automobile Sathish Kumar K Rajamanickam C S 978-1-4673-6150-7/13, IEEE
[3] Experimental and Analytical investigation of the effect of body kit used with saloon cars M.G.Yazdani, H. Ullah, T. adries and R.Zainulariffin
[4] Aerodynamic Study of Formula SAE Car, SnehHetawala,*, MandarGophaneb, Ajay B.K.c, Yagnavalkya Mukkamalad, science direct, 97(2014) 1198-1207
[5] CFD study of section characteristics of Formula Mazda race car wings , W. Kieffer, S. Moujaesand N. Armbya science direct, 43(2006) 1274-1287.

