www.ijmer.com Vol.2, Issue.3, May-June 2012 pp-1189-1194 ISSN: 2249-6645

Optimum Way to Increase the Fuel Efficiency of the Car Using Base Bleed

Prof. G Sivaraj*, Mr. M Gokul raj**

*(Department of Aeronautical Engineering, Bannari Amman institute of technology, Sathyamangalam, India)
** (Department of Aeronautical department, Bannari Amman Institute of technology Sathyamangalam, India)

ABSTRACT

Ever since then the invention of car have undergone several modifications in terms of speed, comfort, fuel efficiency, and other features In this paper we shall glance to reduce the drag force of car using base bleed .In this injects low velocity air into the rear side of car region. Computational analysis was carried out using Gambit and Fluent software. As per the computational analysis it's proved that, the drag co-efficient of car model was reduced. In other word car spends the least power in overcoming the drag exerted by air and hence exhibits higher performance- cruises faster and longer, that too on less fuel.

Keywords: **Fuel efficiency, Drag force, Computational Analysis, Base Bleed, Car Model**

I. INTRODUCTION

Aerodynamic styling of a car is one of the most crucial aspects of car design-a highly complex phenomenon, encompassing the task of an artful integration of advanced engineering and stylish aesthetics. A lot of emphasize is laid on the aerodynamics in car design as an aerodynamically well designed car spends the least power in overcoming the drag exerted by air and hence exhibits higher performance- cruises faster and longer, that too on less fuel.

Apart from improved fuel economy, aerodynamically superior car offers better stability and handling at highway speeds and also minimization of harmful interactions with other vehicles on the roadway

Consequently, in the present era of enormously soaring prices of fuels with rapidly exhausting resources, and growing awareness among the consumers with regard to safety and other offered features, optimization of car aerodynamics, more precisely the reduction of associated drag co efficient (C_D), which is mainly influenced by the exterior profile of car has been one of the major issues of the automotive research centers all around the world. Average C_D values have improved impressively over the time, from 0.7 for old boxy designs of car to merely 0.3 for the recent more streamlined ones.

Aerodynamics is basically the study of how easily air glides over the surface of car. Air while moving past the car exerts two different forces on car surface,

- Tangential forces induced by shear stresses due to viscosity and velocity gradients at boundary surface, and
- Forces normal to the car surface resulting from pressure intensities varying along the surface due to dynamic effects.

Vector sum of theses tangential and normal forces integrated over complete surface gives a resultant force vector. Component of this force in the direction of relative velocity past the car body is known as aerodynamic drag.

Aerodynamic drag, which compares the drag force, at any speed, with the force it would take to stop all the air in front of the car influences fuel consumption of a car, especially at higher speeds and hence is considered a crucial factor in judging its performance. An aerodynamically well designed car spends the least power in overcoming the drag and hence yields higher performance - cruises faster and longer that too on less fuel.

The forces of wind resistance against car are called drag. We measure a car's ability to slip through the wind by assigning it a drag coefficient (C_D), which is calculated through a mathematical equation. The C_D equation is

$$C_D = D / (0.5\ \rho\ V^2 A),$$

where , D is drag,

ρ is air density,

V is velocity, and

A is the car's frontal area.

Reducing drag and lowering that C_D is one of the main goals of aerodynamic improvements, as getting a car to slice cleanly through the wind has several benefits. Probably the most important one is efficiency. Reduce the drag that is acting on a vehicle and you can reduce the power needed to push it through the air. An engine that does not have to work as hard gets better fuel economy, which is why aerodynamics are so important to the new car industry.

Over the top of the car, the air is also initially decelerated and again at the foot of the windscreen which means that pressure is tending to increase. But once over the highest point of the car, the favourable pressure gradient accelerates the air, so the pressure drops.

II. METHOD INVOLVING OF DRAG REDUCTION

Designers have come up with all sorts of tricks to lower drag, from reducing a car's frontal area to molding

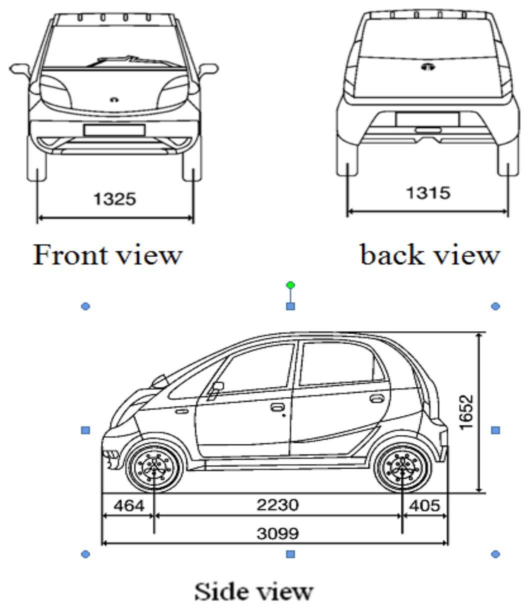
www.ijmer.com Vol.2, Issue.3, May-June 2012 pp-1189-1194 ISSN: 2249-6645

in small winglets under the trunk area to diffuse the air that comes out from underneath it.

But drag is just one aerodynamic factor that's at work on car. Two other forces we have probably heard about before are lift and down force. Look at the side view of a modern car and the shape looks something like the cross-section of an airplane wing, there could be enough lift to unload the tires and suspension, which will affect the car's traction and handling.

To investigate the aerodynamic performance of a car, computational analysis have been carried out using Gambit and fluent software.

III. DIMENSIONS OF CAR USED FOR ANALYSIS



IV. FABRICATION OF BASE BLEED AND ITS PRINCIPLE

BASE BLEED

Base bleed is two converging hollow tube which attached above the bottom of the car. Front end base bleed cross section is larger than the rear end so it injects low velocity air into the rear side of car region. This flow is known as base bleed. Which will results in reduction of the drag co-efficient.

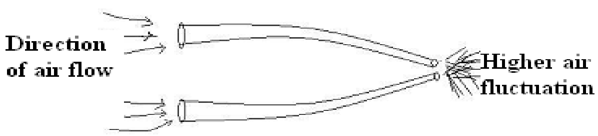


Fig. 1 Air flow of Basebleed

The main function of base bleed is

- Reduces the drag at low speed of the car itself (ie) from 50 kmph
- Increase the rear side pressure and reduce the front side pressure which intern reduces the overall drag of car.
- Reducing wake region at the rear side
- It improves the aesthetics of car as well as it improves the aerodynamics of car.

V. BASE BLEED ATTACHMENT

The nose features a single opening for the front grille and side air intakes, with aerodynamic sections and profiles designed to direct air to the coolant radiators and the new flat underbody. The nose also sports small aero elastic winglets which generate down force and, as speed rises, deform to reduce the section of the radiator intake and cut drag.

Technical development of the car's shaped started using CFD (Computational Fluid-Dynamic) techniques which helped optimize the different cross section of base bleed and interaction of the internal flows prior to wind tunnel testing.

The air intakes for engine bay cooling are situated on the aerodynamic underbody, where differences in pressure channel the air in the most efficient manner, and are positioned to increase rear downforce. Similarly air is channeled from the front air dam to the rear diffuser where the position and number of the fences has been developed to optimize the distribution of the vortex to improve rear downforce.

VI. DIFFERENT TYPES OF BASE BLEED CROSS SECTION

The base bleed has analyzed three different cross section of its. The three kinds of cross section are

- rectangle
- circle
- Ellipse

a) CIRCULAR CROSS SECTION

Frontal cross section

- circle
- radius = 4.068 mm
- area = 52mm^2

Ending cross section

- circle
- radius = 1.5 mm
- area = 7.069 mm^2

a) ELLIPTICAL CROSS SECTION

www.ijmer.com Vol.2, Issue.3, May-June 2012 pp-1189-1194 ISSN: 2249-6645

Frontal cross section

- ellipse
- A = 6 mm
- B = 2.75mm
- area = 52mm2

Ending cross section

- circle
- radius = 1.5 mm
- area = 7.069 mm2

b) RECTANGULAR CROSS SECTION

Frontal cross section

- rectangle
- A = 13 mm.
- B = 4mm
- area = $52mm^2$

Ending cross section

- circle
- radius = 1.5 mm
- area = $7.069\ mm^2$

c) LOCATION OF BASE BLEED IN CAR

Figure shows base bleed is located in the car model. It made up of rubber material and it is flexible.

Fig. 2.Location of Base bleeds in car model

VII. COMPUTATIONAL APPROACH

In the computational approach, data concerning three-dimensional flow field around the body of car was visualized by simulating the flow conditions using Gambit as the preprocessing software and Fluent as the solver and post processor.

PRE PROCESSING

a). GEOMETRY CREATION

The analysis of car aerodynamics can present a significant challenge, requiring the simulation of many different configurations and positions of both car and attachment of base bleed. Wind tunnel analysis with a rolling road is often impractical. The deployment of CFD Fluent 6.1 and Gambit within the design process, however, enables such studies to be carried out with relative case.

When air flows over the surface of a car, a boundary layer forms where there is a large velocity gradient. In order to capture this phenomena correctly, the mesh around the surface of the body must be very fine. To perform this boundary layer study, I will be creating and solving two different meshes. All of the mesh parameters will be staying the same between the two meshes except for the boundary layers.

b) DEFINE THE GEOMETRY

When the geometry was defined in the creation of the computational mesh, all faces of the domain were assigned names. The names of the inlet and outlet planes (at x = 0 and x = L) are front face and back face of domain as velocity inlet and pressure outlet respectively. The names of the planes at y = L, z=0, and z=L are outer wall as wall. The names of the model are car as a wall. And bottom face is defined as road.

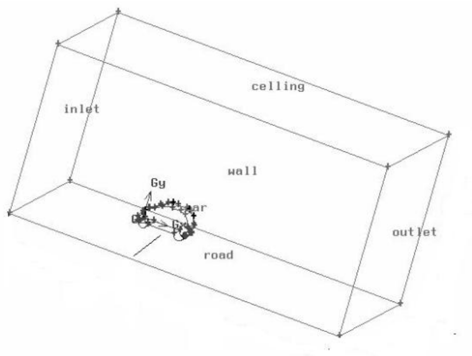


Fig. 3.Defining the geometry

Dimensions of the Domain are

Height `	=	20 m
Length	=	35 m
Breath	=	20 m

c). MESHING

An inflated boundary of prismatic elements was used near the car surface to improve spatial resolution and gain a better understanding of boundary layer phenomena. An unstructured mesh with polyhedral elements was used for volume meshing. Simulations were carried out with the turbulence model, coupled with a blend factor of 0.5 for the advection scheme.

The computational mesh was constructed automatically using polyhedral cells mesh, surrounded at solid boundaries by three prismatic extrusion layers. Because polyhedral cells fill space more efficiently than tetrahedral elements, fewer cells were required than might otherwise have been needed, significantly aiding the goal of using a small desktop machine to perform such aerodynamic analyses

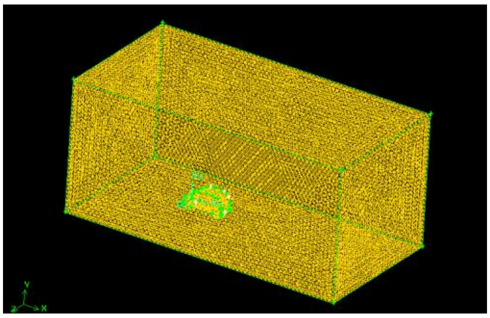


Fig. 4 .3D view of final polyhedral mesh with volume source visible around the car

A model that has already been meshed and it has only 130’000 polyhedral cells. Note that at least 5 million cells, with hexagonal in the near-wall regions, would be necessary to obtain reliable and detailed results in such a case.

The computational domain extends far upstream of the car where the boundary condition will be a velocity inlet. The top and bottom of the computational domain are "periodic" boundary conditions, which mean that whatever flows out of the top goes directly into the bottom. It is assumed that since the outer limits of the computational domain are so far apart, this car behaves as if in an infinite free stream. In order to adequately resolve the boundary layer along the car wall, grid points will be clustered near the wall. Far away from walls, where the flow does not have large velocity gradients, the grid points can be very far apart. A hybrid grid will be used in this problem. Grid adaptation within the flow solver, Fluent, will increase the grid density even more near the wall and wherever else needed.

d). IMPORTING THE MESH

Fluent reads the grid with about 130’000 cells from gambit file. Grid Check is sure there is no negative volume or face area and there is no warning of any kind.e

a. DEFINE THE PHYSICAL MODEL

Define the model of Solver is Segregated for Continuity equation is first solved for all cells, then Momentum and then turbulences. This works well for incompressible and moderate compressible flow. Applying the Implicit for each equation is solved for all cells together with actual dates. The implicit solver brings faster convergence.

Define the model as 3D and Steady. It is Absolute there is no moving mesh zone in the mesh. Define the Model is Viscous as k-epsilon for a robust and efficient turbulent model which gives good results in most cases where turbulences have an isotropic repartition. Define the model of energy equation.

b. SPECIFY MATERIAL PROPERTIES

Define the materials is air And it is properties of

Density = 1.225 $kg \cdot m^{-3}$

Viscosity = $1.464e^{-5}$ $kg \cdot m^{-1} \cdot s^{-1}$

Those values correspond to the ICAO norm. Fluent means dynamic viscosity as we consider air as incompressible and are not looking for heat transfer problematic, we don’t need to specify properties.

e). SPECIFY THE BOUNDARY CONDITIONS

OPERATING CONDITIONS

Let the 101325 Pa which corresponds to the ICAO-Norm. Fluent works with relative pressure.

VIII. BOUNDARY CONDITIONS

Car model is “wall” with “car” (in the field “Zone Name”).We considers our model as a wind-tunnel model. So the car is a stationary wall, the viscosity makes the air stick at the car coachwork, so no slip the coachwork is very smooth, so a roughness of zero. Ceiling of the wind-tunnel and Side wall of the wind-tunnel are specified shear for this will allow the air to slip on the ceiling wall. This is not realistic, but so, we can use a very coarse mesh without boundary layer problems.

Road is specified as Moving Wall. As the car doesn’t move, the road will have a velocity in the positive x-direction, so that the flow under the car will be correctly modeled. Velocity is 25 $m \cdot s^{-1}$ in the Speed field. Correspond to 90km/h. and 0.05m in the “Roughness Height” field.

Inlet is 25 $m \cdot s^{-1}$ in the “Velocity Magnitude” field as the car doesn’t move, the air has to in the positive x-direction. Outlet is Zero Pa in the “Gauge pressure” field means we have atmospheric pressure at the outlet.

a) POST-PROCESSING

Normally we would have to enable better numerical schemes (2nd or 3rd order and run until a much better convergence of the flow solution is reached, but this would take about 3 hours with this case and about 2 weeks with an adequate mesh refinement). So we simply visualize the actual results.

www.ijmer.com Vol.2, Issue.3, May-June 2012 pp-1189-1194 ISSN: 2249-6645

IX. FLOW PARAMETERS TAKEN FOR STUDY

Five important results were obtained from the analysis.

Calculate the co-efficient of drag.

1. Drag force variation along the car model.
2. Static pressure variation along the car model.
3. Total pressure variation along the car model.
4. Path line of velocity magnitude variation along the car model.

All of the above results are analysis car with Base bleed attachment.

X. CONTOUR PLOTS (FILLED) FOR BASE BLEED

Initially three different cross section of base bleed are taken into consideration for analysis, which is circular, elliptical and rectangular.

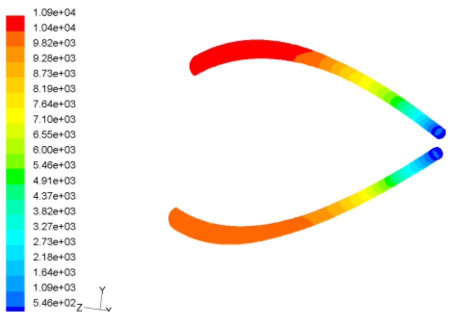


Fig. 5 Static pressure of elliptical base bleed

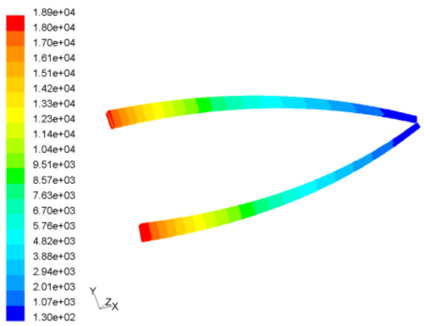


Fig. 6.Static pressure of rectangular base bleed

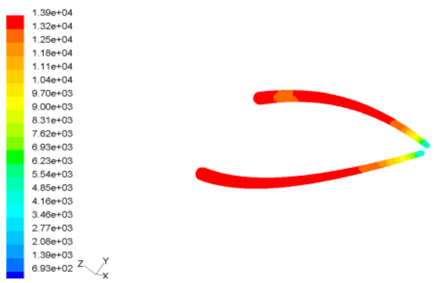


Fig. 7.Static pressure of circular base bleed

XI. STATIC PRESSURE VARIATION OF CAR AT FRONT SIDE

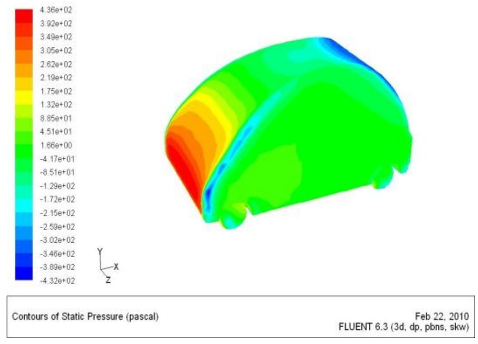


Fig. 8.Car model without base bleed

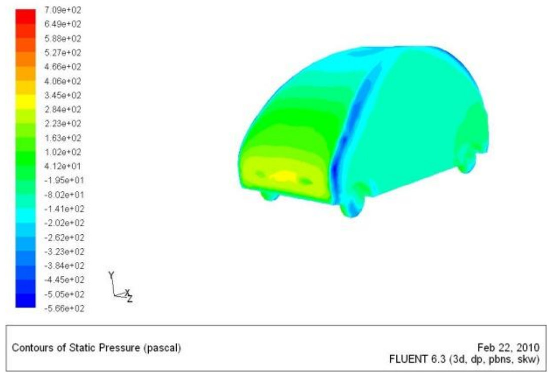


Fig. 9.Car model with base bleed

Three different cross section of base bleed are taken into consideration for analysis, which is circular, elliptical and rectangular. From analysis, it is clearly picturised that, elliptical cross section has higher fluctuation compared to that of other ones, which is clearly shown in above figures. Therefore elliptical cross section Base bleed is attached in the car model for experimental and computational work.

Three dimensional car model with and without Base bleed are taken into consideration for analysis. From analysis it is validated that, three dimensional car model with Base bleed has higher fluctuation compared to that of without Base bleed, which clearly shown in above figures.

XII. RESULTS AND DISCUSSION

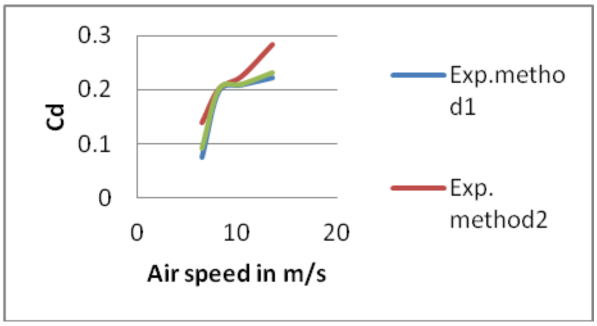
Pressure distribution matches with prediction that pressures would be low in the regions with streamlined profiles such as nose, base of the windshield etc. Almost identical nature of graphs of variation of pressure co efficient along car profile at different air velocities also verified that pressure co efficient is independent of speed

www.ijmer.com Vol.2, Issue.3, May-June 2012 pp-1189-1194 ISSN: 2249-6645

Table 1. Co efficient of drag on car model

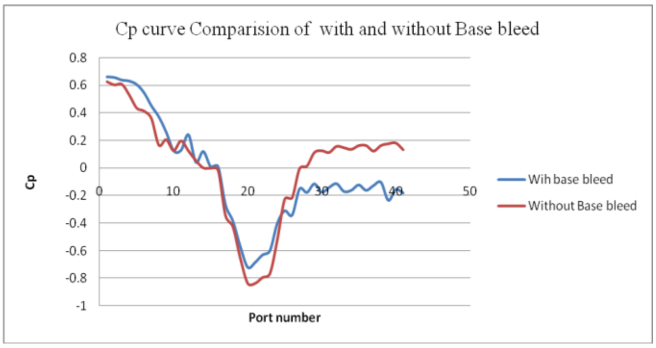
Air speed	Computational result	
	Without Base bleed	With Base bleed
13.6m/s (1000 rpm)	0.3521	0.2321
10.52m/s (750 rpm)	0.3274	0.2109
8.23m/s (500 rpm)	0.2741	0.2013
6.52m/s (300 rpm)	0.1527	0.0927

Using this Table, variation of Co efficient of Drag C_D with air velocity (Graph. a and Graph.b) are plotted



Graph: 1 Variation of Air speed with C_D

Graph shows drag co efficient variation with air velocity of with and without base bleed. Pressure distribution matches with prediction that pressure would be low in the regions with streamlined profiles such as nose, base of the windshield etc., and almost identical nature of figs of variation of pressure co efficient along car profile at different air velocities also verified that pressure co efficient is independent of speed. Computational predictions of external aerodynamics of car showed quite a good agreement with and hence validated experimental once.



XIII. CONCLUSION

Keeping base bleed at the car model it will increase fluctuation energy of the air. As a result of verifications, it is confirmed that base bleed creates stream wise vortices. From this, we could predict that base bleed causes the pressure of the vehicle's entire rear surface to increase therefore decreasing drag.

So it is concluded that base bleed which gives very least drag and also regarding position, after front shield which has a very good effect in delaying the flow separation and also reducing wake region.

ACKNOWLEDGMENT

We first thank our 'GOD', the supreme power for giving us a good knowledge and our parents for making us study in a renowned college We owe a great many thanks to my colleagues and friends for their help and encouragement.

REFERENCES

[1] Fukuda, H., Yanagimoto, K., China, H., Nakagawa, K., "Improvement of vehicle aerodynamics by wake control," *JSAE Review* 16:151-155, 1995.

[2] Gaylard, A.P., "The Appropriate Use of CFD in the Automotive Design Process," *SAE Technical Paper 2009-01-1162, 2009.*

[3] Tanner, M., "Reduction of Base Drag," Prog. *Aerospace Sci. 16(4):369-384, 1975.*

[4] Mair, W.A., "Drag-Reducing Techniques for Axi-Symmetric Bluff Bodies," *AIChE Symposium Series: 161-187*, 1978.

[5] Oertel, H., Jr., "Wakes Behind Blunt Bodies," *Annual Review of Fluid Mechanics 22:539-564*, 1990.

[6] Viswanath, P.R., "Flow Management Techniques for Base and Afterbody Drag Reduction," Prog. *Aerospace Sci. 32:79-129*, 1996.

[7] Morel, T., "Effect of base cavities on the aerodynamic drag of an axisymmetric cylinder," Aeronautical Quarterly 30:400-412, 1979.

[8] Goodyer, M.J., "Some experimental investigations into the drag effects of modifications to the blunt base of a body of revolution," *Institute of Sound and Vibration, University of Southampton, Report No. 150*, 1966.

[9] Nash, J.F., Quincey, V.G., Callinan, J., "Experiments on two-dimensional base flow at subsonic and transonic speeds," *Aeronautical Research Council R & M 3427*, 1966.

[10] Khalighi, B., Zhang, S., Koromilas, C., Balkanyi, S.R., Bernal, L.P., Iaccarino, G., Moin, P., "Experimental and Computational Study of Unsteady Wake Flow Behind a Bluff Body with a Drag Reduction Device*," SAE Technical Paper 2001-01-1042*, 2001.