Drag Reduction in An Automobile by using Aerodynamic Shapes

Arvindakarthik K S
Department of Aeronautical Engineering
Sri Ramakrishna Engineering College
Coimbatore-641030, Tamil Nadu, INDIA

Mohan Kumar K K
Department of Aeronautical Engineering
Sri Ramakrishna Engineering College
Coimbatore-641030, Tamil Nadu, INDIA

Gokul Sangar E
Department of Aeronautical Engineering
Sri Ramakrishna Engineering College
Coimbatore-641030, Tamil Nadu, INDIA

Abstract **- This paper deals with the reduction of drag. The maximum concerns of the automotive aerodynamics are reducing drag, reducing 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 desirable downwards aerodynamic forces to improve traction and thus cornering abilities. The characteristic shape of a road vehicle is bluff, compared to an aircraft and it operates very close to the ground, rather than if free air. The operating speeds are lower. Finally, the ground vehicles has fewer degree of freedom of motion is less affected by aerodynamic forces. Drag reduces the speed of the automobile and also reduces the efficiency considerably at high speed. Therefore, to improve the efficiency and the overall stability of the van a duct can be formed at the top front of the van. By attaching the duct setup, the flow pattern of the air is modified by reducing the vortex. This method is very economical and also compact nature of the van is undisturbed i.e., size remains the same. The drag coefficient of the van can be reduced to the greater extent. This can be implemented as the real time project and improve the overall efficiency of the van.**

Keywords – Drag; Aerodynamics; Drag coefficient; Vortex.

I. INTRODUCTION

Until the mid 1960's [1] the understanding of vehicle aerodynamics was very limited and the shape of the van was left in the hands of the designer. The vans were "boxy" and had large frontal areas.

The growth of computing technology in the 1970's and 80's [1] allowed van manufacturers to analyze air flow around vans and recognize the importance of aerodynamics in vehicle development. This progress allowed further research to be focused on reducing the drag coefficient. It has been shown [2] that 40% of the drag coefficient is dependent on the external shape of the van and is concentrated at the rear of the geometry. It has been established [3] that a systematic aerodynamic study of the various parts of a vehicle can help improve its aerodynamics. Also, computational techniques of estimating airflow aided with comprehensive experimentation can produce sustainable results. Researchers [2] have also investigated the effects of utilizing various geometries for the components of the airfoils behind vans. Others [4] have investigated the wing/body interaction in generic shapes of closed-wheel race-cars. There have also been investigations in the patterns of airflow in case of convertible cars [5]. However, there still great potential for research in this application of fluid dynamics.

A moving van experiences an increase in aerodynamic force with an increase in its velocity. Just like an aerofoil, body of a van experiences drag and lift forces, the only major difference being that due to the shape of a van it experiences a negative lift or down force [6]. The down force generated by its shape gives the van the ability to go around non banked curves with higher speeds. Hence an aerodynamically superior van design generates a low drag force but a high down force. An aerodynamically superior van design can reduce fuel consumptions and increase its efficiency. This produces a more environmentally friendly van. However due to the competitiveness of the motor sport industry, there is limited current research findings pertaining to the aerodynamic features of race car. Subject of this paper is to investigate and analyze the airflow around model van using commercial ANSYS software package.

II. NOMENCLATURE

C_d = Coefficient of drag.

ρ = Density of air.

d = Drag force.

v = velocity of air.

S = surface area.

ISSN: 2278-0181
Vol. 4 Issue 10, October-2015

III. THEORY

The basics of aerodynamics are Bernoulli's equation which state that at any point in a streamlined flow. If the local air stream is lower than that of the undisturbed flow, the loss is compensated by increase in pressure [1]. As the body of the model van is not streamlined, it causes local disturbances in the air flowing around the body. The fluctuation in the local air stream velocity reduces a spectrum of pressure on the various spaces of the vehicle. This pressure creates the rise in forces acting along the three axes [1]. Drag forces is given by d.

The relative velocity of the air mass to the van is the same as the relative velocity of the ground to the van. Hence it is assumed that the velocity of the air mass is equal to the relative velocity of the ground to van. In this investigation, Cd value is assumed to be a constant since the general airflow is turbulent.

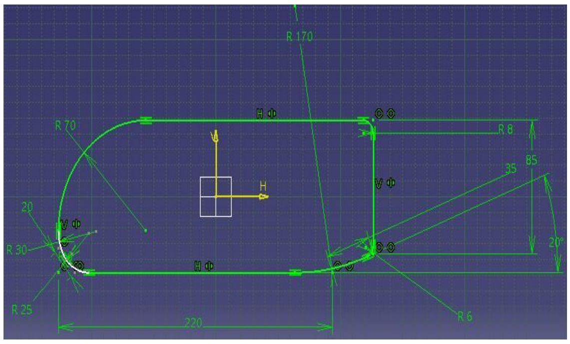
Similarly, for the force given by, And force caused by winds, side force is given by, As shown [6], the flow of the air in the boundary layer of the van has been found to be turbulent for the most part. The boundary layer causes particle stagnation around some region along their body of the van. This effect is known "Skin Friction" [6]. Skin Friction is attributed to the viscous force which significantly alters the flow of air around the van. Hence it is essential to consider viscous forces and model the viscosity of air while analyzing the air flow around the van. Therefore, model viscosity the Navier stoke equation were incorporated into the ANSYS analysis. And formula used to find Cd values are $Cd = (2 * d) / (\rho * v * v * s)$.

IV. METHODOLOGY

A velocity test was varied out in the laboratory to obtain the average velocity of the model van. The velocity test ensured that the boundary condition (VIN) used in the ANSYS model replicated the actual velocity of the model van. A commercial ANSYS software package was used to analyze the airflow around the modern van .The body of the van was scanned using a 3-D scanner to with a precision of 1mm i.e. 0.233%.The scanned data was imported to CATIA and checked for errors and self intersecting geometries .The scanned model and the entire geometry was exported to the ANSYS pack

The ANSYS model did not incorporate the airflow around the wheels and assumed the body of the van to be a shell without wheel arches. This design 1 also assumes no side, up or down winds.

A redesigned van body is used to further analyze and compare the aerodynamic effects of a change of body shape. This redesigned van body has the same vital dimensions i.e. length, width, average height, ground clearance and the volume of the model van.



DESIGN 1

V. THE ANALYSIS

The model vans was analysed for the average velocity which was obtained over 3 straight runs. The velocity analysis was varied out in a laboratory conditions by using ANSYS software. Due to the analysis being varied out indoors the cross wind effect was neglected. The results are shown in table1. The front end of the van experiences significantly higher pressures than the rest of the van. This can be attributed to its poor curvature and high projected area. It is also seen that the pressure head on the rear aerofoil of the van does not generate the down force that is required from the aerofoil. Therefore the inclusion of the rear aerofoil apart from its aesthetic value has no significant aerodynamic contribution to the model 2 van. It demonstrates that the velocity of the air reduces as it reaches the front of the van.

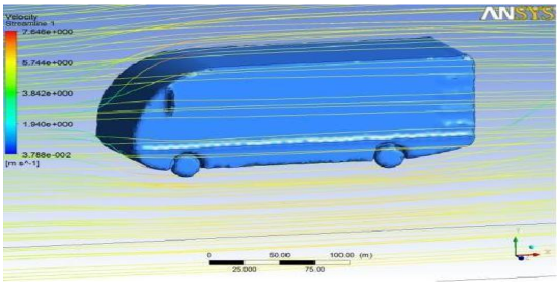
This velocity then increases due to a positive slope above the front end of the van. The airflow velocity over the roof reduces significantly and remains constant as it flows towards the rear of the van. The ANSYS analysis on the model van has shown that the rear airfoil, overhanging arches and frontal shape of this model 1 and model 2 van has contributed to its ineffective aerodynamics performance.

Demonstrate that the redesigned model 3 van has a reduced pressure head in the front end. This pressure head greatly contributes to the down force generated in the front half of the van therefore provides for stability. As the air flows beyond the front grille of the van, the static pressure reduces with an increase in the gradient of the bonnet. This graduated change in the pressure contours reduces the concentrated pressure heads along the upper surface of the redesigned model 3 van.

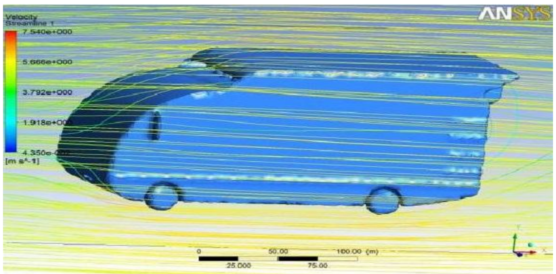
TABLE 1

Model	Lift KN	Drag KN	Cd X10^-5
Model 1	-40	70	5.50
Model 2	-72	80	5.44
Model 3	-50	55	4.79

The reshape contours of static pressure for the redesigned model 3 van front on the redesigned model van acts like a flow tripper [7] delaying the formation of a potentially stagnant boundary layer. This is demonstrated when comparing.The velocity vectors of the airflow over the redesigned model 3 van another essential change made along the side of the van is the elimination of protruded side curves that acted as surfaces of high static pressure. The resigned van has side surfaces that are streamlined i.e. shaped like a fish which allow for a smoother transition of air. Overhanging curve the reshaping of the rear end of the van has been greatly influenced by a designing technique called “bob-tailing” [6]

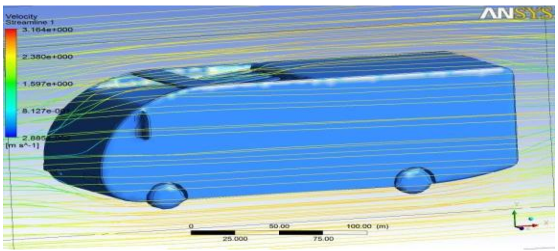


MODEL 1



MODEL 2

The wake i.e. re-going of air recirculation is at the back of the van. In this region both models experience high turbulence and reverse velocities. However “bob tailing “the back reduces the boundary layer stagnation and greatly changes the flow pattern behind the van. This causes a reduction in the wake region which should contribute to a decrease in the drag force.



MODEL 3

VI. CONCLUSION

This investigation has demonstrated that there is a significant change in the coefficients of lift and drag of the model vans when a more streamlined body design is adopted. The results obtained show a reduction in a drag coefficient. This further verifies previous research [8-11] indicating that a body having an “inverted tear drop” shape generates a lower drag force than one with “conventional” shape. The findings of this investigation also confirm the previous research [12-14] detailing that “bob-tailing” the rear end of the van reduces the recirculation region behind the van. This further reduces the drag force acting on the van. Furthermore it was also determined that varying the angle of the front grille has an effect on the down force acting on the van.

VII. FURTHER WORK

Suggestions for further work are:

a) Verifying the computational results by building and testing both in a wind tunnel.

b) Building a practical model of the redesigned van and varying out experimental tests on it to verify the obtained computational results.

c) Gathering data on the actual environmental conditions that the van is exposed to and using that data in the ANSYS analysis.

REFERENCES

[1] Wolf-Heinrich Hucho “Aerodynamics of Road Vehicles “Cambridge Butterworthys, 1987, ch. 1-4

[2] W.Kieffer ,S.Moujaes, N. Ambya “CFD study of section characteristics of formula Mazda race car wings “March 200

[3] F. Muyl, Laurent Dumas, Vincent Herbert “hybrid method for aerodynamics shape optimization“, June 2003

[4] G.de Vahl Davis and C. Fletcher, “computational fluid dynamics “.Amsterdam: Elvesier Science Publishers B.V., 1988, PP 315-328

[5] Hiroshi china, Masahiro yoshida, Morihiro takada, Kunio nakagawa,”air flow control around the cabin of a convertible car” July 1994

[6] Aj scribor –rylski “Road Vehicle Aerodynamics “London: pentech ressl, 1975, ch-1-7

[7] Masaru koike, Tseunehisia Nagayoshi, Naoki hamamoto,”Research on aerodynamic drag reduction by vortex Generators “

[8] Appupillai Baskaran and Ahmed Kashef,”Investigation of air flow around buildings using computational fluid dynamics techniques “, 1996.

[9] Heller, A.”Der neue Kraftwagen von Dr.Ing Rumpler “, 1921

[10] Eppinger, A. “Tropfenwagen- Anwending der Flugzueg Aerodynamik”, 1921

[11] Rumpler, E. “Das Auto im Luftstrom “, 1924

[12] Stapleford, W.R and Vanr.”Aerodynamic characteristics of exposed rotating wheels “1970

[13] Morelli, A. “Aerodynamic actions on an automobile “1969

[14] Loghton, C.J. “Rotating wheel experiment “, Unpublished, City University, London, 1969.