

A Review on - Aerodynamic Drag Reduction of a Passenger Vehicle by Controlling the Wake using CFD Analysis

Jacob K. Musa¹ Prof. P. D. Patel² Sanat M. Gameti³

^{1,3}M.E. Student ²Associate Professor

^{1,3}Department of Mechanical Engineering ²Department of Automobile Engineering

^{1,2,3}L D College of Engineering, Ahmedabad

Abstract— Automobile manufacturers are competing intensely to improve the fuel efficiency of road vehicles. Drag is the highest aerodynamic force encountered by ground vehicles at normal highway speeds. The major contributor is the after-body because of the drag produced by the separation of the boundary layer at the rear. This increases the work done by the engine to propel the vehicle and consequently increase fuel consumption. Among other things reduction of drag is important for better fuel economy and vehicle handling.

Key words: CFD, Wake, Aerodynamic drag, Wind tunnel, Drag coefficient, Drag Reduction

I. INTRODUCTION

Aerodynamics makes its major impact on modern cars and trucks through its contribution to “road loads”. Aerodynamic forces interact with the vehicle causing drag, lift (or down force), lateral forces, moments in roll, pitch and yaw, and noise. These impact fuel economy, handling and NVH (Noise –Vibration-and Harshness). The aerodynamic forces produced on a vehicle comes from two sources – form (or pressure) drag and viscous friction. [10]. With the current increase in fuel prices together with the new policies such as Paris Summit 2015 on fossil fuel emissions have brought into focus its significance of better fuel economy and less global warming. Growing concerns of future security of fuel supplies and depletion of fossil fuels have led to long term understanding of the implications of fuel use. This has motivated many scholars to analyze airflow over passenger ground vehicles in order to reduce aerodynamic drag and increase fuel efficiency and reduce harmful emissions. With such motivation, scholars in the field of engineering have come up with different modeling techniques and software packages to analyze flow over bodies in order to get the best results for reducing drag.

II. EXISTING LITERATURE

Fukuda Group [1]: attempted the improvement of vehicle aerodynamics by wake control. Their results showed that the rear end spoilers on sedans reduce rear aerodynamic lift, but they tend to increase aerodynamic drag. They then developed a spoiler to reduce rear aerodynamic lift without increasing aerodynamic drag.

Amaruddin Z. Maazouddin and Dongmei Zhou [2]: Studied drag reduction on SUVs and Trucks by Wake Control. Gillieron and Chometon’s [12] computer Ahmed Reference Model was selected as a benchmark test for the computer model and simulation of SUVs validation. The model and simulation were reproduced in their study by following the outline explained in “Modelling of Stationary Three – Dimensional Separated Air Flows around an Ahmed Reference Model”. Seven different spoiler designs for the SUVs were studied. The results for the SUV’s aerodynamics

were presented through velocity vectors, pressure contours, and aerodynamic lift and drag plots. The results showed that there was only one successful design of the spoiler which reduced drag. The spoiler design also reduced the size of the turbulent wake region, the drag coefficient, but it increased the aerodynamic lift coefficient. It was also concluded that the reduction in the turbulent wake region would lead to better fuel economy and the decrease in aerodynamic lift would lead to better vehicle stability. The objective is to reduce aerodynamic drag acting on the vehicle and thus improve the fuel efficiency of passenger car. Hence, the drag force can be reduced by using add on devices on vehicle and fuel economy, stability of a passenger car can be improved.

Bahram Khalighi, S. Zhang, C. Koromilas, S.R. Balkanyi, Luis P. Bernal, G. Laccarino, P., Moin [3]: Researched on Experimental and computational investigation of a drag reduction device for bluff bodies in ground proximity. The research was done to gain better understanding of the drag reduction mechanisms in bluff-body Square-back geometries using CFD. The detailed investigation of the flow field in the near wake of a square – backed bluff body with and without a drag reduction device was carried out. The sharp edges in the basic SB Model (without tail plate) produced a flow separation which prevents the pressure from recovering, therefore inducing high pressure drag. The addition of the device MSB Model (with tail plates) resulted in reduction of total drag by 20 per cent. Furthermore, the MSB Model generated higher base pressure compared to SB. CFD simulations showed that the drag reduction device strongly influences the unsteady base pressure and velocity spectral peaks at a strouhal number of 0.07 suggesting that, for the present geometries, the drag reduction device suppresses large – scale turbulent motions in the wake. The results also show a reduction of the turbulence intensity as well as a rapid upward deflection of the underbody flow due to the add-on- device compared to the baseline configuration.

Mitsui Okada, John Sheridan and Mark Thompson [4]: In the Study of the “Effect of width – to – Height Ratio on wake structures of simplified vehicle Geometry”, it was demonstrated in their work that the rear slant angle was not the only main geometric parameter that affects the rear wake and wake structure behind a simplified vehicle model (Ahmed Model). It was concluded that the ratio of the width and height has also to be considered when discussing the critical geometry.

Vishal Shukla, Gaurav Saxena [5]: analyzed “Computational Drag Analysis of Passenger Car Using Splines and Spoiler”. The geometry of the car base model was created in CATIA and Simulation & Testing of base line passenger car was done using CFD analysis. First Simulation & Testing of passenger car with Splines was

done and later with a Spoiler for drag coefficient & lift coefficient calculation. This was in view to find the impact of add on devices on fuel economy of passenger car. In the case of splines and the rear spoiler the coefficient of drag is 0.51 and the coefficient of lift is 0.22. The percentage reduction in drag coefficient in comparison of base line car is 15% and in coefficient of lift is 12%. Hence drag force & lift force on the passenger car is reduced as proportional to drag coefficient and lift coefficient respectively. The effects of different aerodynamic add-on devices on flow and its structure over a generic passenger car may be analysed using CFD approach.

Vivek K., and Dr. S. S.Mantha [6]: Studied "Prediction of aerodynamic drag of a car with image based modeling and CFD simulation". The geometry of the car models were created in Proe (wildfire 2.0). For maintaining the accuracy, real-time photographs were traced on graph papers. The following conclusion was made from the analysis: In general the Squareback design suffers from a large recirculation region behind the body due to flow separation, which results in higher drag coefficients owing to lower base pressures at the rear. In case of Fastback designs the flow is complex due to interaction of the longitudinal vortices and separation in the rear. The slant angle of the fast back determines the characteristics of the flow. The Notchback design of car bodies refers to the three box version which has a boat tail ramp to accommodate streamlines behind the body and minimal shedding of vortices and thus avoid separation. Such designs have a low value of drag coefficient owing to high pressure recovery at the base. External car Aerodynamics has been one of the most interesting & challenging CFD applications. Most of the predictions and calculations in this area are obtained by finite volume numerical code based on Reynolds averaged Navier-Stokes (RANS) equations as modeling equations which illustrate CFD capabilities & predict accurate, reliable solutions, (Accuracy -2.70 % with a 95 % confidence band of 13.1. This is in view of the third approach constituted by CFD which augments pure theory & pure experiments.

Gaurav kumar Tandel, Mayur Makwana, Jigar Patel [7]: Performed experimental analysis on a Maruti Esteem car scaled down wooden model 1:20 Scale. From the results of the wind tunnel, the pressure distribution and drag force for different air velocity concluded that high pressure zone is created in the front of radiator part of model and this pressure increases with velocity increase. Also, the drag force increases with increase in air velocity. Negative pressure created at the back side of the car increased with increasing air velocity that indicated the suction in back side of car which creates wake region at the rear end of the vehicle.

Pankaj Madharia, Mau Mohan Tiwari, and K. Ravi [8]: Studied Computational simulation analysis of Ahmed Body with varying Nose radius, Ground height and Rear Slant angle at 250 and 350. The nose radius was varied at R100mm and R120mm. Modification on ground clearance from 50mm to 60mm was carried out. The results compared at inlet velocity of 40m/s showed a reduction in drag as the nose radius increases from base value for both rear slant angle. It was also found, that there was large wake formation of low intensity with the increase in slant angle,

which resulted in the dissipation of energy. Increasing the ground clearance has a counter effect on coefficient of drag. Ravindra A. Viveki and Dr. N. K. Cougule: [9]: Analysed flow over a MIRA reference car by optimizing roof curvature for drag reduction. Modification in roof curvature was done on the baseline model and three parameters – velocity, thickness to chord ratio and position of thickness to chord ratio were used for the optimization purpose to reduce drag using Taguchi method and air flow was simulated. Results showed that as roof curvature increases from the mid position of roof, wake zone reduces hence vehicle drag coefficient reduced by 2.58%. It was also observed that roof curvature will lead to more low pressure zone above roof and this will lead to more lift and was found to be less than 1%.

III. CONCLUSION

The following are the concluding remarks made from the reviewed literature:

- Separation zone occurs at the rear and it is in this area that the maximum potential for drag reduction is possible.
- Controlling the rear wake would lead to delay in the separation of the boundary layer which could be done by using add- on devices.
- Drag reduction methods on a car have so far reached saturation and therefore passive methods such as (add – on devices) shows a promising ability to reducing drag without making major changes to the critical body profile of a passenger car.
- CFD is a vital tool for vehicle aerodynamic analysis as they are cost effective, more information available and faster than wind tunnel experiments.
- Among the passenger cars the squareback car suffers the most from high amounts of base pressure drag because of separation of boundary layer.

REFERENCES

- [1] Fukuda, Hitoshi, Yanagimoto, Kazuo, China, Hirashi, Nakagawa, Kunio, "Improvement of Vehicle Aerodynamics by Wake Control", Society of Automobile Engineers of Japan Review, Vol.16, 1995, pp. 151-155.
- [2] Amarddin Z. Maazouddin and Dongmei Zhou, "Drag Reduction on SUVs and Trucks by Wake Control", ASME International Mechanical Engineering Congress and Exposition, 2008, pp.1-12.
- [3] Bahram Khalighi, S. Zhang, C. Koromilas, S.R. Balkanyi, Luis P. Bernal, G. Laccarino, P. Moin, "Experimental and Computational Study of Unsteady Wake Flow behind a Bluff Body with a Drag Reduction Device", Society of Automobile Engineers, 2001, pp. 1-15.
- [4] Mitsushi Okada, John Sheridan and Mark Thompson, "Effect of Width – to –Height Ratio on Wake Structures of Simplified Vehicle Geometry", International Engineering Research Journal, (IERJ), 2003.
- [5] Vishal Shukla, Gaurav Saxena, "Computational Drag Analysis of Passenger Car Using Splines and Spoiler", International Journal of Engineering Trends and Technology (IJETT), Vol. 21 No.1, 2015, pp. 1 – 11.

- [6] Vivek k., Dr. S. S.Mantha, “Prediction of aerodynamic drag of a car with image based modeling and CFD simulation”, *International Journal of Advanced Engineering Applications*, Vol.3, Iss.6, 2010, pp.45-57.
- [7] Gauravkumar Tandel, Mayur Makwana, Jigar Patel, “Experimental Aerodynamic Analysis of Automobile Vehicle”, *International Journal for Engineering Development and Research (IJEDR)*, Vol.3, 2014, pp. 259-368.
- [8] Pankaj Madharia, Mau Mohan Tiwari, and K. Ravi, “Computational Simulation of Ahmed Body with Varying Nose radius, Ground height and Slant angle”, *International Journal for Research in Applied Science and Engineering Technology (IJRASET)*, Vol.3, No.5, 2015, pp. 925-932.
- [9] Ravindra A. Viveki and Dr. N. K. Cougule, “Computational Fluid Dynamics Optimization of Roof Curvature for Drag Reduction of Passenger Car”, *International Engineering Research Journal (IERJ)*, Vol. 1, No.5, 2015, pp.224-231
- [10] *Fundamentals of Vehicle Dynamics* by Thomas D. Gillespie, Published by Society of Automobile Engineers, Inc, 1988, pp.78-103.

