EXPERIMENTAL AND COMPUTATIONAL DRAG ANALYSIS OF SEDAN AND SQUARE-BACK CAR

Bhavini Bijlani¹, Dr. Pravin P. Rathod², Prof. Arvind S. Sorthiya³

Address for Correspondence
¹PG Student, ²Associate Professor, ³Associate Professor Department of Mechanical Engineering, Government Engineering College, Bhuj-370 001

ABSTRACT
The main aim of this paper is investigation of aerodynamic of sedan and square-back car, measuring drag coefficient and air flow around the car body. There are two methods used for investigation of drag force simulating the air flow by CFD and by using wind tunnel experiment. 1:20 aluminum scale model of popular sedan and square-back car is used for experiment. Experiment is done on subsonic wind tunnel which test section is (30cm x 30cm x 100cm). Computational analysis was carried out in ANSYS CFX-13.

KEYWORDS — Drag force, sedan, Fastback, Wind-tunnel, Aerodynamic CFD.

INTRODUCTION
A variety of studies of the aerodynamic influence of vehicle rear end shapes have been researched including the study of critical geometry found by Hucho et al. It is well known that the rear-end shape of a car is one of the important elements which governs aerodynamic drag and lift.[14]

Due to fuel cries, researchers attention much of the focuses on lowering the vehicle drag coefficient (C_d), which accounted to about 75 to 80% of total motion resistance at high speed. Airflow over a vehicle determines the drag forces, which in turn affects the car’s performance and efficiency testing equipment has been designed to measure both the vertical and horizontal components of air resistance on a model car[6].

However, as the passenger car need enough capacity to accommodate passengers and baggage so there is minimum necessary space for its engine and other components. It is extremely difficult to realize an aerodynamically ideal body shape. The car have a body shape that is not perfect an ideal streamline shape as seen fish and birds. Such a body shape is inevitably accompanied by flow separation at the rear end[1]. Two major element that have major influence on the drag coefficient of a bluff object are the roundness of its front corners and the degree of taper at its rear end[1].

Aim of this paper is to investigation of aerodynamic of sedan and Square-back car, by experimentally and computationally.

EXPERIMENTAL APPROACH
Wind tunnel is a tool used in aerodynamic research to study the effects of air moving past solid objects. The wind tunnel used in the present work is subsonic type with a squared test section of 30cm×30cm with a glass window meant for visual observation of flow phenomenon and maximum velocity 30m/s. A provision for traversing Pitot tube in horizontal direction (Fig.3) was created to meet with specific requirement of suggested experimentation methodology.

Experimental investigations in wind tunnel Tests is to be conducted on a geometrically similar, reduced scale (1:20) aluminum model, differing from actual car only in size and simulating dynamically similar flow situations.

Fig-2 Test Section

The model instrumented with pressure tapings along the centre line, over its profile, was tested at different air velocities. The Static pressure distribution was obtained and represented in terms of a non dimensional parameter pressure obtain pressure distribution on the centre line of the car.

Fig-3 Alluminum model

Fig-4 Pressure Distribution Along The Centre line at a Particular Air Velocity (Sedan)

\[
F_d = \sum P \cos \theta \cdot A
\]

Where \( \theta \) is angle between direction of relative velocity and normal pressure force.
III COMPUTATIONAL APPROACH

The CFD Solver uses the Reynolds Averaged Navier-Stokes equations expressed in 3-D form. To avoid unnecessarily long computations was modeled geometrically on the CFX pre processing plate form and define control volume and generating mesh (Fig-6).

A 3D steady-state, incompressible solution of the Navier-Stokes equations was obtained by implementing turbulent modeling with standard k-ε model using standard wall functions and second order upwind discretization scheme[10].

The flow field data was generated in the post processing part of the analysis in CFX and contours of static Pressures (Fig-7) and velocity Vectors (Fig-8) were obtained and analyzed in detail. The values of drag is calculated by the post processing were employed to validate the plots obtained by the experimental procedures.

IV RESULT AND DISCUSSION

Drag force of all car model are increases as speed increases. Sedan has a low drag force as compare to square-back car. There is correlation between experimental values and computational values are represented graphically for sedan and square-car is shown in figure-12 and figure-13 respectively. Computational value of Drag force of Sedan car has higher value than obtained experimentally by 11% Where as drag of Square-back car computationally has higher value than obtained experimentally by 12% respectively.

Distribution of Pressure obtained experimentally is in agreement with prediction that distribution of The pressure is negative over nose, the back light and on to the trunk because of the continuing curvature or the streamline shape. The front radiator zone has higher pressure contours. The pressure contours by CFD at different velocity are identical with pressure contour obtained by experimentally and thus validate the experimental procedure and result.

The investigation showed that the sedan car produce better aerodynamic compared to square-back as drag coefficient for sedan is lower than at about 42%. Sedan is more streamline compare to Square-back as the air flows for square-back detached from the car surface earlier than sedan. The value of drag coefficient ($C_d$) is to be around 0.38 for sedan and 0.66 for square-back experimentally.

CONCLUSION

Experimental approaches for the investigation of external aerodynamics of a 1:20 scale car model of sedan and square-back car were presented. Agreement between values predicted by both experimentally and
computationally confirms their reliability and recommends them for further experimentation. The drag coefficient ($C_d$) evaluated for exterior profile of Sedan and Square-back, to be of the order of 0.38 and 0.66, which are acceptable. Sedan is more aerodynamic than square-back. However, further optimization of this wind tunnel estimation is strongly recommended. Experimental investigations were further validated computationally. Combining wind tunnel experiments and CFD computation, integration on both methods can be lead to a better aerodynamic design.

REFERENCES


5. J.P. Howell "The side load distribution on a Rover 800 saloon car under crosswind conditions" Journal of Wind Engineering and Industrial Aerodynamics 60 (1996) pp. 139-153


8. Prof.P.R Sonawane, Prof.S.P.Sekhawat, Prof .K.Rajput "Aerodynamic analysis of car body for minimum fuel consumption journal of information knowledge and research" in mechanical engineering ISSN 0975-668X volume-01, issue-02, Nov-10 to Oct 11, pp. 54-57.


13. Hitoshi Fukuda, Kazuo Yanagimoto, Hiroshi China, Kenji Nakagawa,

