COMPUTATIONAL VALIDATION OF EXPERIMENTAL AERODYNAMIC PREDICTIONS OF A CAR

MANAN DESAI, S. A. CHANNIWALA, H. J. NAGARSHETH
Mechanical Engineering Department
Sardar Vallabhbhai National Institute of Technology (SVNIT)
Surat, Gujarat
INDIA
madrene23@gmail.com, sac@med.svnit.ac.in, hjn@med.svnit.ac.in

Abstract: - In the recent times, CFD simulations [1], with the advent of computer architectures with superfast processing capabilities are rapidly emerging as an attractive alternative to conventional wind tunnel tests which are either too restrictive or expensive, for aerodynamic styling of a car. The paper describes validation of experimental aerodynamic predictions by conventional wind tunnel approach with advanced computational procedures, carried out as a part of design process of a small car proposed to be named as ADRENe. The experimental investigations were performed on an open circuit suction type wind tunnel having a 30cm x 30cm x 100cm test section, on a geometrically similar, reduced scale (1:15) clay model, while the three dimensional computational analysis was carried out using Gambit as the preprocessing software and Fluent as the solver and post processor. The comparison shows that the computed drag forces and pressure distributions agree well with the experimental values over the entire range of air velocities, however, the agreement with the data for drag coefficient varies, which appears to suggest a higher degree of dependency on the grid quality and elements selection.

Key-Words: - Car design, Aerodynamics, Wind tunnels, CFD

1 Introduction
In car design-a highly complex phenomenon [2], encompassing the task of an artful integration of advanced engineering and stylish aesthetics, a lot of importance is paid to aerodynamic styling as an aerodynamically superior car exhibits higher performance and also offers better maneuverability, stability and minimization of harmful interactions with other vehicles on the roadway, typically at high speeds[3].

Aerodynamics is basically the study of how easily air glides over the surface of a car. Air while moving past the car exerts two different forces on car surface [4], i) Tangential forces induced by shear stresses due to viscosity and velocity gradients at boundary surface; and ii) Forces normal to the car surface resulting from pressure intensities varying along the surface due to dynamic effects. Vector sum of these 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 performance of a car is usually predicted in terms of drag coefficient (CD), which basically compares the drag force, at any speed, with the force it would take to stop all the air in front of the car. An aerodynamically well designed car would spend the least power in overcoming the drag and hence would yield higher performance - cruise faster and longer that too on less fuel [5]. The recent, modern streamlined designs of cars have CD values in the range of 0.3 to 0.5[6].

The paper describes experimental investigation of aerodynamics of a small car ADRENe, as a part of its design process, and its subsequent validation with computational techniques. The performance was predicted experimentally by resorting to the most reliable and conventional [7] wind tunnel tests while computationally in the environments of Fluent.

2 Experimental Approach
In the conventional mode, the aerodynamics of the proposed exterior profile of ADRENe was investigated in an open circuit, suction type wind tunnel [8, 9] having a 30cm x 30cm x 100cm test section with a Plexiglas window meant for visual observation of the flow phenomenon. A variable speed DC motor employed varies the air velocities in the range of 5 to 40 m/s. A provision for traversing the Pitot tube in horizontal direction was created specially [Fig.1] to meet with the specific requirement of the suggested experimentation methodology.

A geometrically similar, reduced scale (1:15) [10], clay model, differing from the actual car only in size was employed for the testing purpose as it would be enormously expensive to engineer and build a full scale car [11] for wind tunnel testing just to find out if the
nearest and the farthest edges as inlet velocity and exit pressure respectively [Fig.6]. All car surfaces and the floor were treated as walls and no-slip wall conditions were applied.

A 3D steady-state, incompressible solution of the Navier-Stokes equations was obtained by implementing turbulent modeling with standard $k-\varepsilon$ model using standard wall functions and second order upwind discretization scheme. The free stream air velocities for a series of tests were varied from around 10m/s to 40m/s while the exit pressure was set to 1.013bar.

The flow field data was generated [18] in the post processing part of the analysis in Fluent and contours of static pressures and velocity profiles, supplemented by path lines [Fig.7] were produced and analyzed in detail to make some initial judgment about the aerodynamic performance of the car.

The critical value of drag coefficient (CD) comes out to be around 0.4 experimentally and around 0.55 by simulations, which might be due to the fact that a simplified model is being analyzed computationally as has been justified earlier. Had all the geometrical details / features influencing the aerodynamics been incorporated in the simulated model, the margin could have been further decreased. However, the values in any case are falling well within the permissible range, confirming acceptance and promising an efficient performance of the proposed profile.

4 Results & Discussions

Computational predictions of external aerodynamics of ADRENe showed quite a good agreement with and hence validated experimental ones.

Variation of drag force (FD) with air velocities and initial decrease of drag coefficient (CD) and then attainment of almost a constant value with further increase in Reynolds number [Fig.8,9], was very much in confirmation with theoretical understanding [19,20].

Pressure contours demonstrating high pressure zones in the front of radiator portion and low in the regions with streamlined profiles such as nose, base of the windshield etc. (Fig. 7) matched well with the nature of distributions obtained theoretically.
4 Conclusions

Experimental investigations of external aerodynamics of a small car ADRENe were validated computationally. The performance was predicted experimentally by resorting to the most reliable and conventional wind tunnel tests, which were extremely time consuming. However, the computational procedure adopted, due to the benefit of advanced computer configurations with high speed processing capabilities, could rapidly generate data for the entire flow field around the car, providing significant insight into the basic flow features, made the evaluation process faster and cost effective.

The comparison shows that the computed drag forces and pressure distributions agree well with the experimental values over the entire range of air velocities, however, the agreement with the drag coefficient varies, which appears to suggest a higher degree of dependency on the details included in the geometric modeling, grid quality and elements selection. However, the critical value in either case is falling well within the permissible range, confirming acceptance and promising an efficient performance of the proposed profile of ADRENe.

References:


