Experimental and Computational Aerodynamic Investigations 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 comparative assessment of two distinct experimental strategies of aerodynamic predictions by conventional wind tunnel approach and its subsequent validation with advanced computational procedures, carried out as a part of design process of a small hybrid 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.
A good agreement between performance values obtained independently by two experimental methods, suggests their reliability and suitability for further experimentation purposes. The comparison with computational approach 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

Aerodynamic styling of a car is one of the most crucial aspects of car design-a highly complex phenomenon [2], encompassing the task of an artful integration of advanced engineering and stylish aesthetics. A lot of emphasize is laid on the aerodynamics [3] 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. (Fig.1) [4]

Apart from improved fuel economy, aerodynamically superior car offers better stability and handling at highway speeds and also minimization of harmful interactions [5] 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 [6, 7] has been one of the major issues of the automotive research centers all

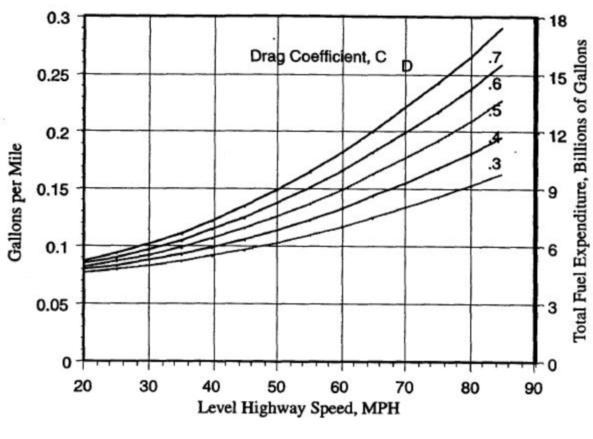

Fig. 1 Fuel economy with reduction in drag co efficient (C_D)

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 [8] 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 [9], 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 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 (AD), 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 (Fig.2) 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.

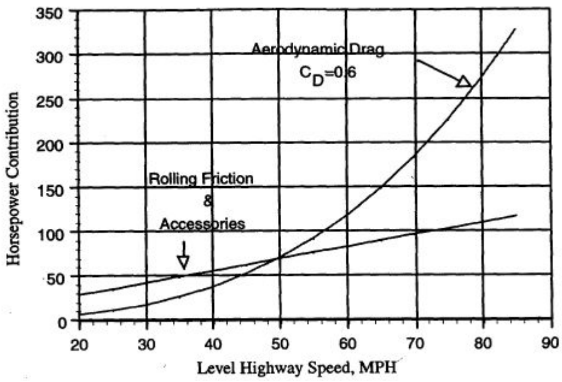

Fig. 2 Increasing influence of drag co efficient (C_D) at higher speeds

The paper describes experimental investigation of aerodynamics of a small car ADRENe and its subsequent validation with computational techniques, to verify acceptability of its proposed exterior profile as a part of its design process.

The performance is investigated experimentally by resorting to the most reliable and conventional [6] wind tunnel approach, by conducting tests adopting two different techniques, one relying on the measurement of pressures in effective domain upstream and downstream of car, and the other on pressure distribution along the centerline over car profile, on a geometrically similar reduced scaled model differing from the actual car only in size and simulating dynamically similar flow situations. The performance is further validated employing commercially available Gambit as the preprocessing software and Fluent as the solver and post processor. [10, 11]

2 Experimental Approach

Experimentally, tests were carried out in an open circuit suction type wind tunnel [12, 13, 14] (30cm x 30cm x 100 cm) with a Plexiglas window meant for visual observation of flow phenomenon. A variable speed DC motor employed varies air velocities (5-40m/s). A provision for traversing Pitot tube in horizontal direction (Fig.3) was created specially to meet with specific requirement of suggested experimentation methodology.

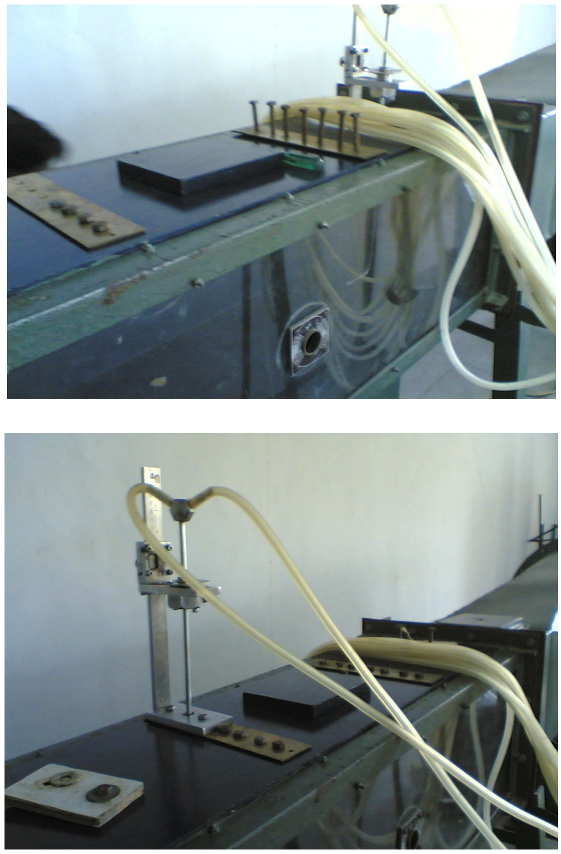

Fig. 3 Experimental investigations in wind tunnel

Tests were conducted on a geometrically similar, reduced scale (1:15) [15], clay model, differing from actual car only in size and simulating dynamically similar flow situations as it would be enormously expensive to engineer and build a full scale car [2,16] for wind tunnel testing just to find out if the design is acceptable.

2.1 Method 1

In this approach of predicting aerodynamic performance, scaled model was placed on the floor of wind tunnel test section (Fig. 4) and air was blown at different velocities.

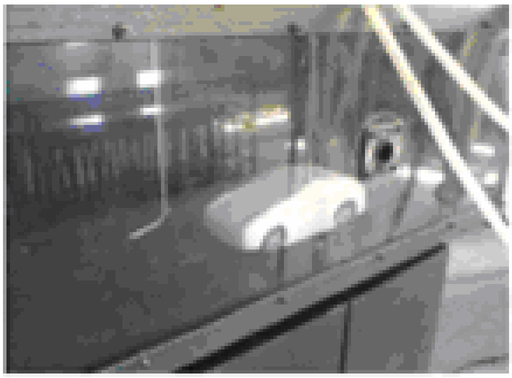

Fig. 4 Experimental set up (Method 1)

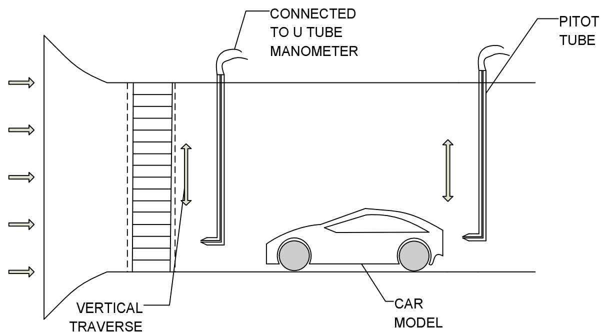

Fig. 5 Pitot tube traversing (Method 1)

The variations in the static pressure at different nodes were measured by traversing Pitot tube (Fig.5) in horizontal and vertical directions, thereby covering entire flow field on upstream as well as downstream sides of car.

For a particular air velocity on upstream and downstream sides,

$$h_{avg,xn} = \sum_{i=1}^{7}\left(\frac{h_{yn,i}}{7}\right) \quad n = 1,2,3$$

$$h_{avg} = \sum_{n=1}^{3}\left(\frac{h_{avg,xn}}{3}\right) \qquad ....(1)$$

$$\text{Drag force } (F_D) = \rho.g.\left(h_{avg,up} - h_{avg,down}\right).A \quad ....(2)$$

Where,
h_{avg} = average height of water column in the manometer limbs
A = car frontal area
While suffixes x and y correspond to the position of pitot in horizontal and vertical directions

Using equations (1) and (2), variation of F_D with air velocity (Fig.6) and C_D in relation with Reynolds number (Fig.7) are plotted.

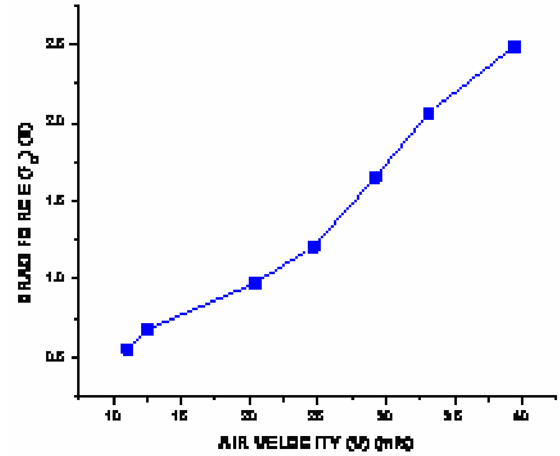

Fig.6 Variation of drag force (F_D) with air speed (V) (Method 1)

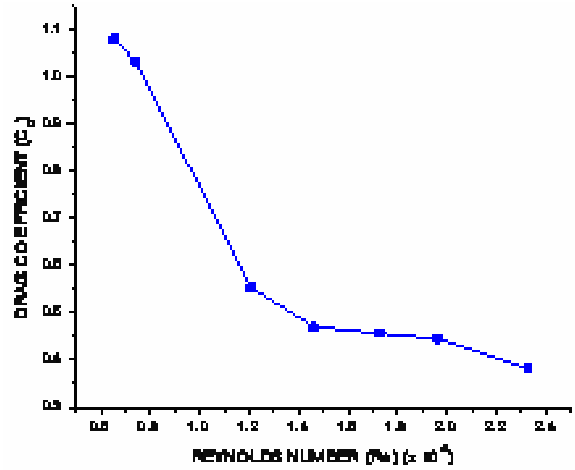

Fig.7 Variation of drag coefficient (C_D) with Reynolds number (Re) (Method 1)

2.2 Method 2

In an another approach, the model instrumented with pressure tapings along centerline, over its profile, was tested at different air velocities (Fig.8, 9).

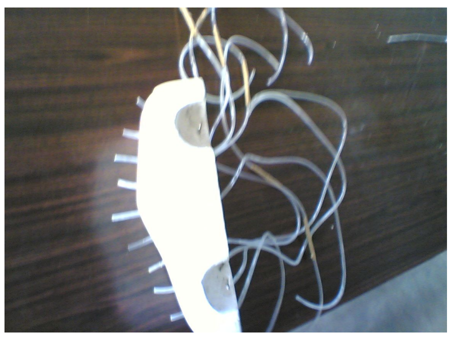

Fig. 8 Pressure tapings

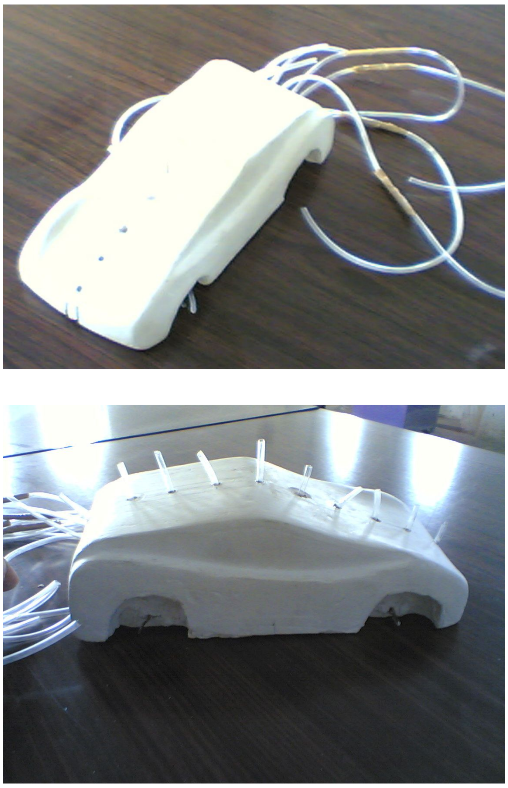

Fig. 9 Model with pressure tapings (Method 2)

Static pressure distribution so obtained [17, 18], was represented in terms of a non dimensional parameter- pressure co efficient (Cp). (Fig.13)

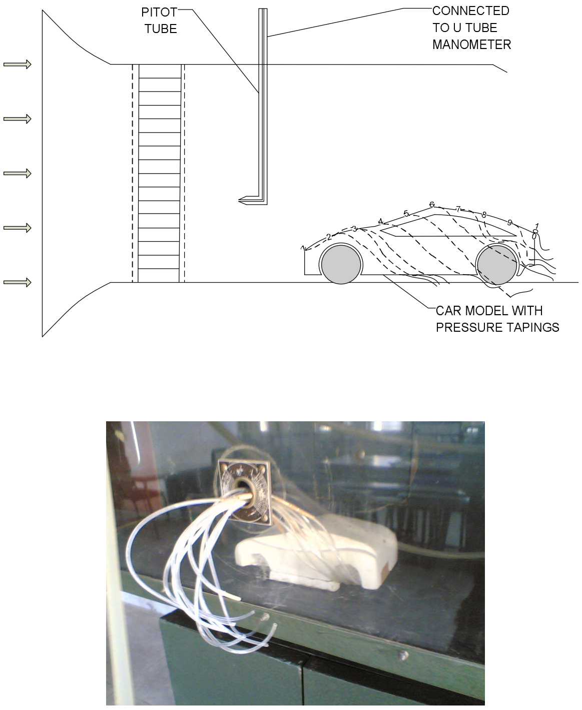

Fig. 10 Experimental set up (Method 2)

Variation of Cp along the centerline at a particular air velocity (Fig.11) and at a particular position along centerline at different air velocities (Fig.12) is plotted.

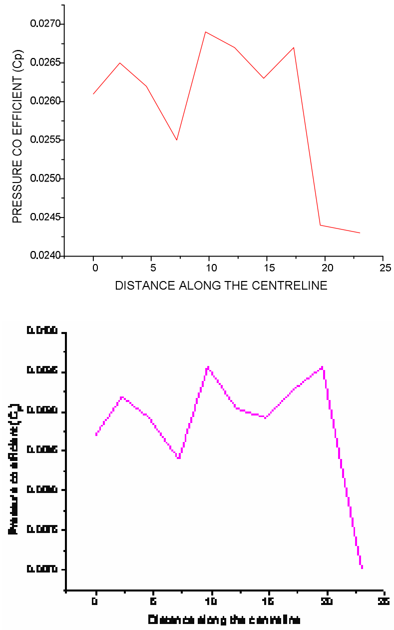

Fig. 11 Pressure variation along the centerline at a particular air velocity

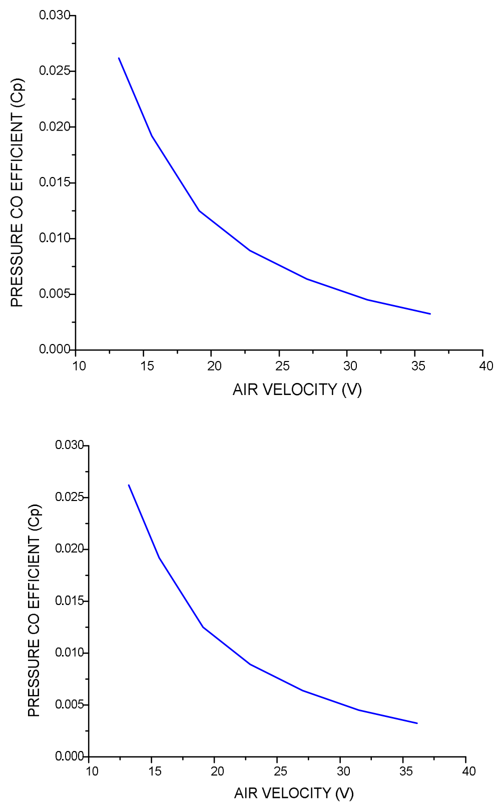

Fig.12 Pressure variation with velocity at a particular location along the centerline

From the pressure distribution obtained (Fig.13);

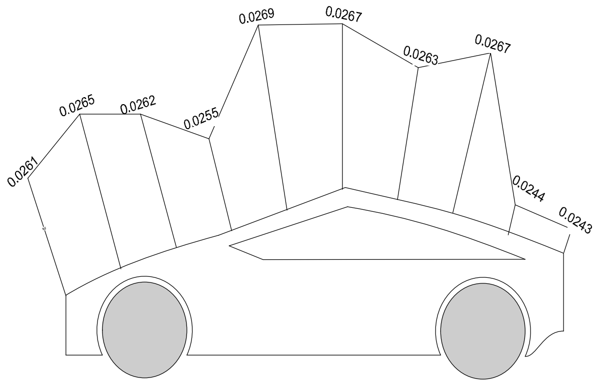

Fig. 13 Pressure distribution over car profile (Method 2)

$$\text{Drag force } (F_D) = \sum_{i=1}^{10} (p_i.\cos\theta_i).A \qquad ....(3)$$

Where θ is angle between direction of relative velocity and normal pressure force

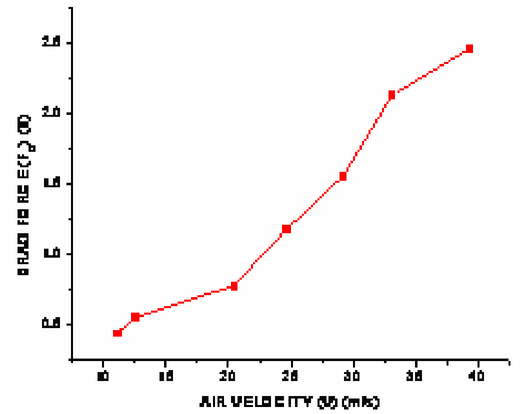

Fig.14 Variation of drag force (F_D) with air speed (V) (Method 2)

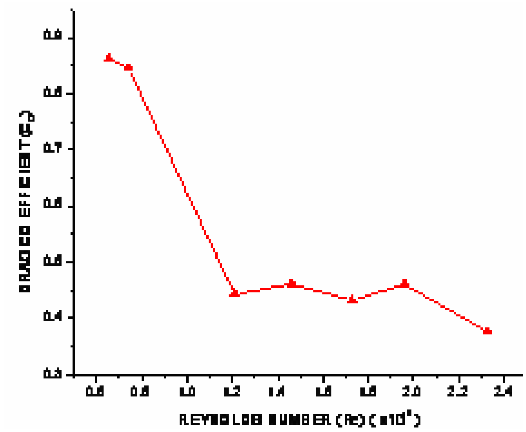

Fig.15 Variation of drag coefficient (C_D) with Reynolds number (Re) (Method 2)

Using equations (1), (2) and (3), similar plots of variation of F_D with air velocity (Fig.14) and C_D in relation with Reynolds number (Fig.15) are plotted for this approach of experimentation as well.

2.3 Computational Approach

In the computational approach (Fig.16), data concerning three-dimensional flow field around the body of ADRENe was visualized by simulating the flow conditions using Gambit as the preprocessing software and Fluent as the solver and post processor.

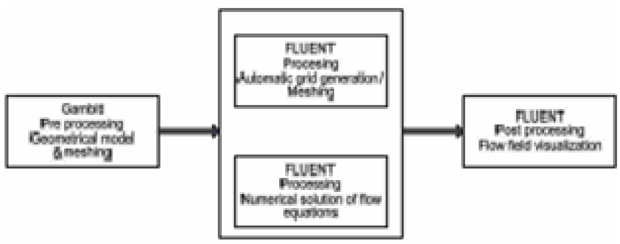

Fig.16 Computational Approach

A simplified model [3], which can efficiently stand out the main facets of the problem, to avoid unnecessarily long computations was modeled geometrically on the Gambit platform (Fig.17). The underbody flow and tires were omitted. All the scaling dimensions, including ground clearance, were properly selected yielding blockage ratio less than 10% [19] and ignorable wall effects.

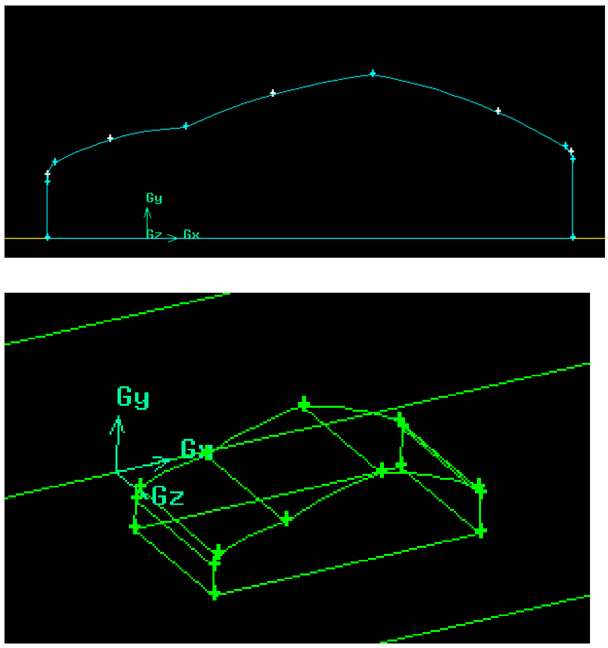

Fig.17 Geometrical Modeling

The grid was generated automatically [20, 21] in the three dimensional, rectangular computational domain (Fig.18) using tetrahedral unstructured mesh cells while the model surfaces were covered with uniformly distributed triangular elements.

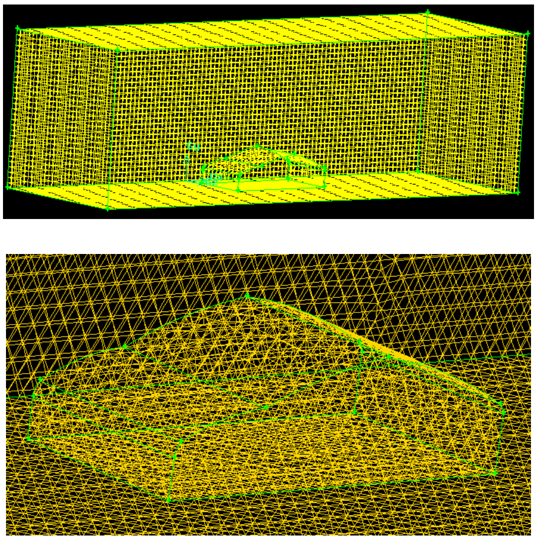

Fig.18 Grid Generation

For flow visualization, as a part of processing in Fluent, boundary conditions were specified on the nearest and the farthest edges as inlet velocity and exit pressure respectively (Fig.19). All car surfaces and the floor were treated as walls and no-slip wall conditions were applied.

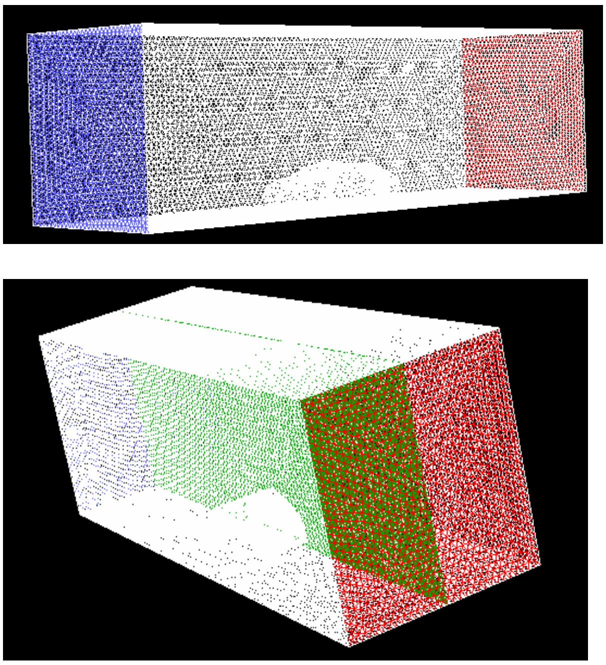

Fig.19 Boundary condition specification

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. 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.

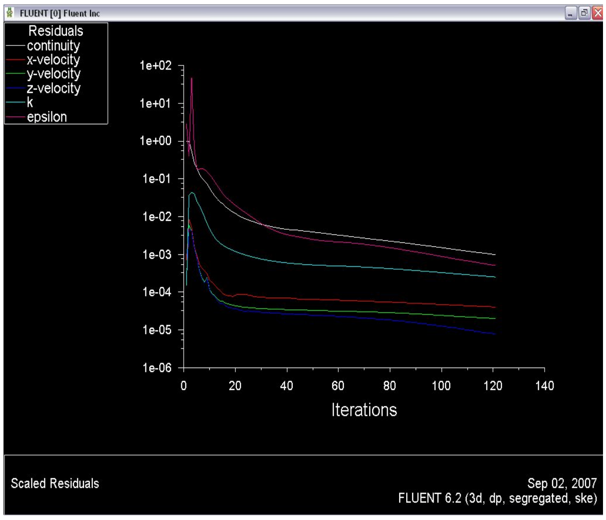

Fig.20 Processing

The flow field data was generated [22] in the post processing part of the analysis in Fluent and contours of static pressures (Fig,21) and velocity profiles, supplemented by path lines (Fig.22) were produced and analyzed in detail to make some initial judgment about the aerodynamic performance of the car.

The values of computed drag revealed by the post processing were employed to validate the plots obtained by the experimental procedures.

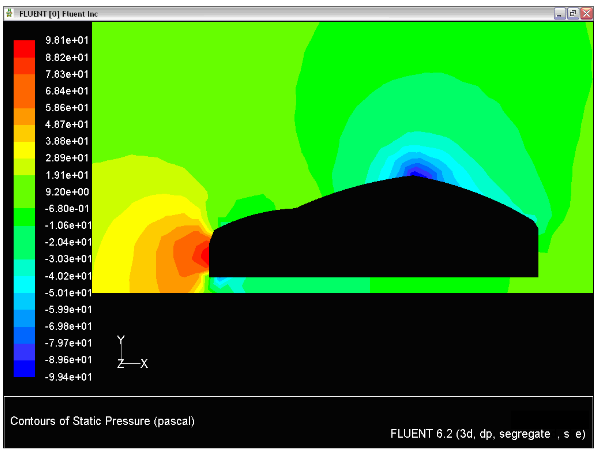

Fig. 21 Pressure Contours

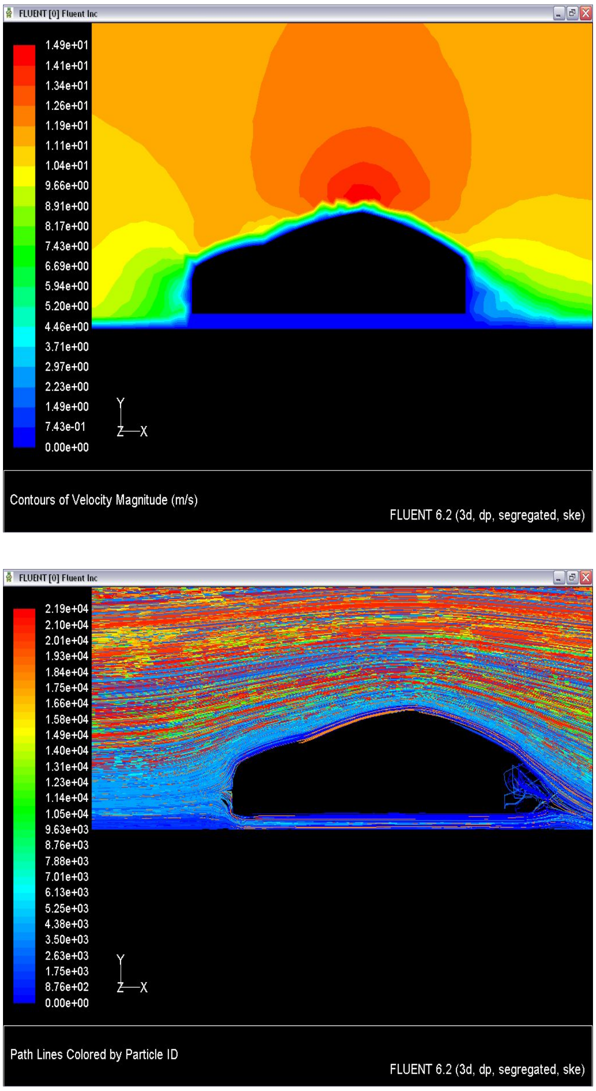

Fig.22 Contours of pressure, velocity and path lines

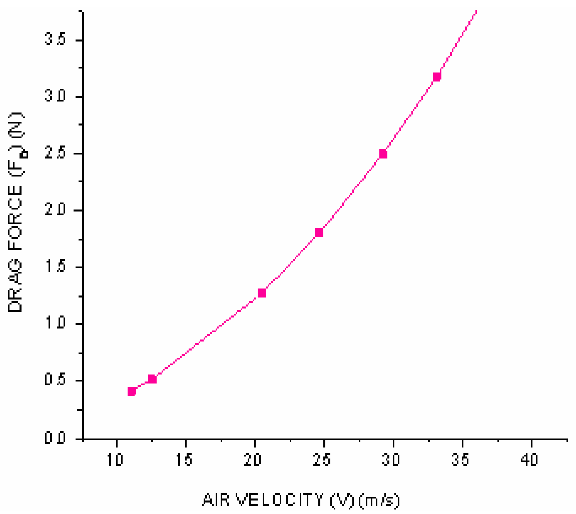

Fig.23 Variation of drag force (F_D) with air speed (V) (FLUENT)

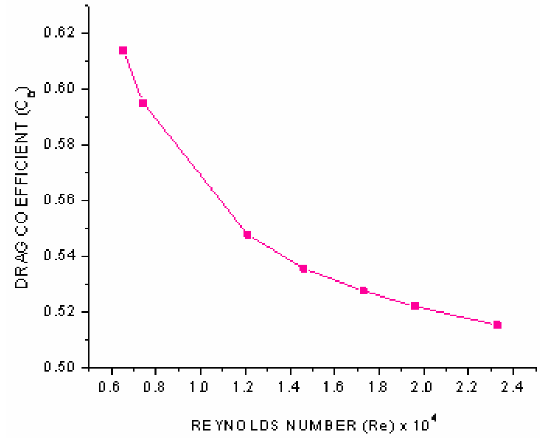

Fig.24 Variation of drag coefficient (C_D) with Reynolds number (Re) (FLUENT)

3 Results and Discussion

Comparison of results (Fig. 25, 26) obtained independently by both the experimental approaches displayed a good agreement. Variation of drag force with air velocities and initial decrease of drag co efficient and then attainment of a constant value with further increase in Reynolds number [Fig.7,15] was very much in confirmation with theoretical understanding [23, 24].

It appears, however, in lower speed range second method estimates the values of drag co efficient on a bit lower side. Critical value of drag coefficient C_D comes out to be around 0.38 corresponding to a Reynolds number of 2.2 x 10^5 and beyond, which meets with expected target, confirming acceptance of the proposed profile and promising an efficient performance.

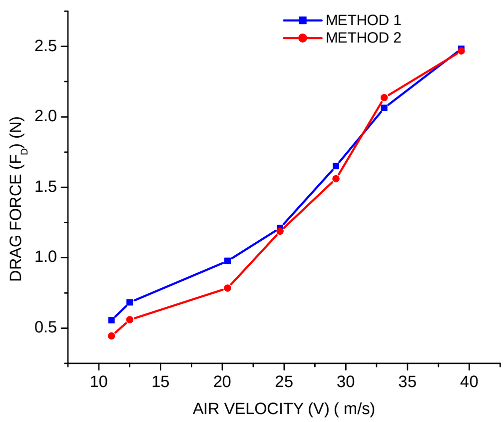

Fig.25 Variation of drag force (F_D) with air speed (V) (Method 1 & 2)

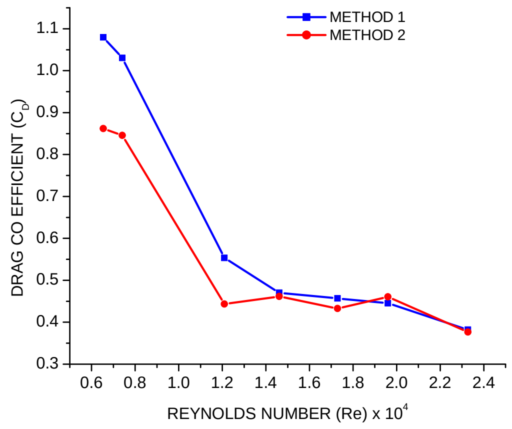

Fig.26 Variation of drag coefficient (C_D) with Reynolds number (Re) (Method 1 & 2)

Pressure distribution [Fig.13] 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 [25] (not included here) also verified that pressure co efficient is independent of speed.

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

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. 21) matched well with the nature of distributions obtained theoretically.

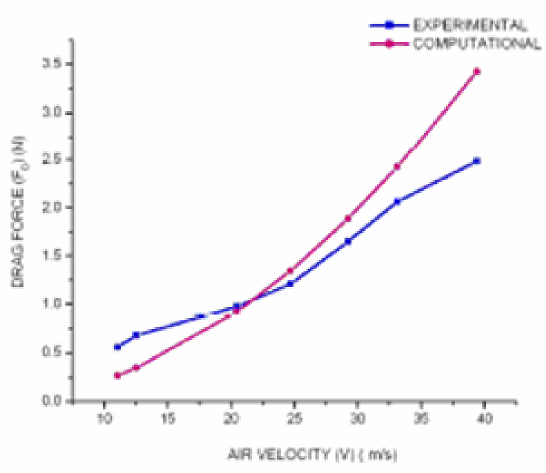

Fig.27Variation of drag force with air velocity (Computational /Experimental)

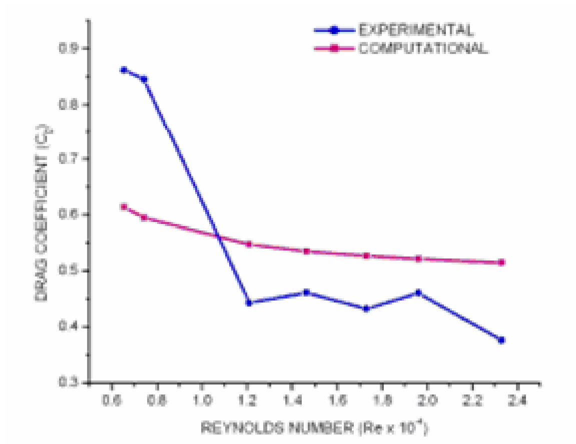

Fig.28 Variation of drag co efficient with Reynolds number (Computational/Experimental)

The critical value of drag coefficient (C_D) 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 Conclusions

Two experimental approaches for the investigation of external aerodynamics of a small car ADRENe were presented. Agreement between values predicted by both methodologies confirms their reliability and recommends them for further experimentation. The drag co efficient (C_D) evaluated for proposed exterior profile of ADRENe, comes out to be of the order of 0.4, which is sufficiently acceptable for the said purpose of effective fuel savings. However, verification and further optimization of these wind tunnel estimations are strongly recommended.

Experimental investigations were further validated computationally. Unlike wind tunnel tests, 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:

[1] W.Kieffer, S.Moujaes, & N.Armbya, CFD Study of Section Characteristics of Formula Mazda Race Car Wings, *Mathematical and Computer Modeling*, Vol. 43, , June 2006, pp. 1275-1287.

[2] H. Braess Hermann & U. Seiffert, *Handbook of Automotive Engineering*, SAE International, Warrendale, SA, 2005

[3] J. Aider, L. Dubuc, G. Hulin & L. Elena, Experimental and Numerical Investigation of the Flow over a Simplified Vehicle Shape, *Proceedings of 3rd MIRA International Vehicle Aerodynamics conference*, (Rugby, UK), 2000

[4] W. Stapleford, Aerodynamic Improvements to the Body and Cooling System of a Typical Small Saloon Car , *Journal of Wind Engineering & Industrial Aerodynamics*, Vol. 9, 1981 pp. 63-75

[5] R. McCallen, F. Browand, A. Leonard, & W. Rutledge, Systematic Approach to Analyzing and Reducing Aerodynamic Drag of Heavy Vehicles, *Annual Automotive Technology Development Customers' Coordination Meeting* (Dearborn, Michigan) October 27-30,1997

[6] F. Muyl, L. Dumas & V. Herbert, Hybrid Method for Aerodynamic Shape Optimization in Automotive Industry, *Computers & Fluids*, Vol. 33, 2004, pp. 849–858

[7] W. Hucho, *Aerodynamics of Road Vehicles*. (London: Butterworth), 1997

[8] J. Katz, *Race Car Aerodynamics: Designing for Speed*, Bentley Publishers, Cambridge, USA, 1995

[9] J. Daily & Donald R. Harleman, *Fluid Dynamics*, Addison-Wesley Publications, 1966

[10] K. Andreas, Simulation of Air Flow around an Opel Astra Vehicle with FLUENT, *Journal Articles by Fluent Software users*, JA132, 2001

[11] F. Alam, A. Subic, J. Naser, M. Rasul & M. Khan, A Study of Spin Effects on Tennis Ball Aerodynamics, *WSEAS Transactions on Fluid Mechanics*, Issue 3, Vol. 3, July 2008, pp. 271-278

[12] D. Sun, H. Wu, C. Lam & R. Zhu , Development of a Small Air Vehicle Based on Aerodynamic Model Analysis in the Tunnel Tests, *Mechatronics*, Vol.16, 2006, pp. 41–49

[13] K. Garry, Wind Tunnel Tests on the Influence of Fixed Ground Board Length on the Aerodynamic Characteristics of Simple Commercial Vehicle Models, *Journal of Wind Engineering & Industrial Aerodynamics*, Vol. 38, 1991, pp.1-10

[14] M. Malerba, M. Argento, A. Salviuolo & G. Rossi, A Boundary Layer Inspection on a Wing Profile through High Resolution Thermography and Numerical Methods, *WSEAS Transactions on Fluid Mechanics*, Issue 1, Vol. 3, January 2008, pp. 18-28

[15] R. Barnard & H. Vaughan, The Aerodynamic Tuning of a Group C Sports Racing Car, *Journal of Wind Engineering & Industrial Aerodynamics*, Vol. 22, 1986, pp. 279—289

[16] R. Shevell, *Fundamentals of Flight*, 2nd Edition, Pearson Education, 2004, pp. 51-58

[17] Lopes & P. Carvalheira, On the Application of Numerical Methods for the Calculation of the External Aerodynamics of a Streamlined Car Body, *SAE Technical Paper*, 2003-01-1249

[18] J. Vogt, T. Barber & E. Leonardi, Flow Field Phenomena about Lift and Downforce Generating Cambered Aerofoils in Ground Effect, *WSEAS Transactions on Fluid Mechanics*, Issue 3, Vol. 3, July 2008, pp. 234-243

[19] Tuve & Domholdt, *Engineering Experimentation*, Prantice Hall, 1985

[20] Y.Sun, G.Wu & Xieshuo, Numerical Simulation of the External Flow Field around a Bluff Car, 2000

[21] E. Nielsen & W. Anderson, Recent Improvements in Aerodynamic Design Optimization on Unstructured Meshes, *AIAA Journal*, Vol.40, No.6, 2000, pp. 1155-1163

[22] R. Himeno & K. Fujitani, Numerical Analysis and Visualization of Flow in Automobile Aerodynamics Development, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol.47, 1993, pp. 785-790

[23] F. Molina-Aiz, D. Valera, A. Alvarez & A. Maduen, A Wind Tunnel Study of Airflow through Horticultural Crops: Determination of the Drag Coefficient, *Biosystem Engineering*, Vol.93, No.4, 2006, pp. 447-457.

[24] R. Fox & A. McDonald , *Introduction to Fluid Mechanics*, John Willey & Sons, New York, 1998

[25] M. Desai, Preliminary Design of a Hybrid Car, PhD Progress Report, 2007