Aarthi S, 2026, 14:2
ISSN (Online): 2348-4098
ISSN (Print): 2395-4752

International Journal of Science, Engineering and Technology
An Open Access Journal

Aerodynamic Drag Reduction of a Passenger Car Model Using CFD Analysis in ANSYS Fluent

Aarthi S, Amirdha Varshni S, Janani Kavya R, Jayabharathi R
Dr. S. Santhanakrishnan
Department Of Mechanical, Meenakshi Sundararajan Engineering College, Kodambakkam.

Abstract- In this world as there is increment growth in automobile industry also there is need to develop a high efficient vehicle which can take high speed performance with high stability. An improvement in high speed of a vehicle depends upon amount of drag force subjected to a vehicle and it has adverse effect of increment in energy consumption of the vehicle. Since the testing using Wind tunnel is expensive and time consuming, it is required to develop some effective numerical methods. The paper states to provide a CFD based aerodynamic analysis of an object which could be useful for determining and reduction of drag force for the passenger car by the application of ANSYS Fluent software. Critical aerodynamic quantities to be observed for such a system are the distributions of pressure, the contours of velocity and the wake and streamlines in relation to a vehicle. To provide an efficient prediction such critical quantities are given with relevant turbulence models and boundary conditions as velocity inlet, pressure outlet and no slip wall condition. Pre-processing for such a problem can be defined as geometry creation, domain creation and meshing. The obtained results based on the experiment show that the redesign of the geometry has reduced the value of coefficient of drag. Such CFD methodology to provide cost effective and efficient car analysis prior to any physical prototype can be implemented in latest car designing.

Keywords: CFD Analysis (Computational Fluid Dynamics), Aerodynamic Analysis, Drag Force Reduction, Passenger Car Aerodynamics, ANSYS Fluent, Numerical Methods.

I. INTRODUCTION

Motor industry is changing dynamically, demanding higher efficiency and better stable driving characteristics in cars. Among all forces, the aerodynamic drag, opposing to the motion when speed increases, is the dominant resistance experienced by vehicle and the minimization of aerodynamic drag on the vehicle is one of the crucial aims of automotive design and development process.

Wind tunnel test is the conventional method for experimental test on the performance of aerodynamics. It has a limitation to perform a costly and time-consuming process with complicated wind tunnel model setup. With the progress of computational technique, CFD (Computational Fluid Dynamics) has become one of the promising tool to investigate aerodynamic flow around automotive bodies in different operation conditions.

Numerical simulation using CFD models permits visualizing flow pressure, velocity contours, stream-lines and the wake region behind the vehicle, all of which contribute to finding the zones of flow separation leading to higher drag. Therefore, modifications on the vehicle geometry is useful to improve aerodynamic performance from simulation. This project is intended to minimize the aerodynamic drag on passenger car model through CFD numerical analysis using ANSYS Fluent software. Geometry modeling, computational domain definition, meshing, boundary condition and solver setup, and post-processing are all performed to predict the variations in drag coefficient. Comparison of the modified vehicle with base model shows a lower drag force with an improved aerodynamic performance. An approach to perform assessment on aerodynamic automotive performance using CFD seems to be cost effective to some extent.

II. LITERATURE REVIEW

Vehicle Aerodynamics is a fast growing research field over the last few decades owing to the continuous need for fuel efficient and performance oriented vehicles. Traditional aerodynamic analysis were always based on experimental methods (Wind tunnel method), the technique being one to predict reliable results for flow characteristics and drag force measurements around a given shape. These methods required rigorous methodology, were expensive and tedious, so not effective for a rapid design optimization cycle.

With the objective to reduce testing time, computational methods (CFD) have been implemented as an alternative to study the external aerodynamics of vehicles. These methods enable numerical simulation of the flow over the complex geometry and allow prediction ofdrag coefficient,pressure distribution,velocity contours, flow visualization around and behind the vehicle,wake structure and locations of flow separation which leads to maximum drag. Extensive work has been carried out by several researchers where adjustments have been made to different aspects of the geometry such as roof shape, rear end shaping, addition of spoilers and smoothness of the under-body which led to minimization of flow separation and thereby reduce the pressure drag in the wake region.

In order to study flow behavior as well as to understand the basics of drag reduction in vehicular aerodynamics before considering full-scale complex geometries, many simplified vehicle models were used. In this thesis work, several turbulence models have been applied to predict the aerodynamic characteristics of a vehicle, regarding its shape under steady state flow conditions and visualization, along with streamline patterns and pressure variation. The above studies helps in shaping the vehicle body with reduced drag coefficient, shows appropriate results for turbulence modeling on flow behavior and reduction in drag, taking several forms of turbulence models into account. However, a lot of research was undertaken directly towards both criteria separately without much effort to combine them together to get a better optimal vehicle aerodynamics. The work herein tackles this problem by designing an aerodynamic simulation-based analysis process with a CFD code called ANSYS Fluent, to predict passenger cars' drag characteristics.

III. EXISTING SYSTEM

The traditional means of assessing the aerodynamic design of a vehicle body have been through the use of wind tunnels and road tests. The values that were measured were: drag coefficient and lift properties, and the flow pattern across the surface of the vehicle. Similar to other fields of engineering, tests on physical models in laboratories dependent on such testing, significantly increases the development cost and test time, along with makes it very difficult to make design changes to the vehicle and re-run testing during the early stages of design. Most traditional automobile body aerodynamic optimisation algorithms are formulated by using approximations based on observation and algebraic estimation of drag reduction performance.

Traditional aerodynamic methods lack any visualisation of the flow around the body of the vehicle. Consequently they fail to have an accurate understanding of complex flows around the body, including; boundary layer separation, and wake regions, and hence the impact of very minor variations in vehicle geometry upon performance is poorly predicted prior to prototype testing. Past work has looked at various body shapes and geometric parameters in an attempt to reduce drag (introduction of spoilers, tapering of the body end, smoothing of underbody), usually based upon educated guesses and testing, although very few tests were used to determine which tests actually confirmed the changes in performance.

As a consequence large development time and cost was attributed to developing the prototype and experimental validation. Initial attempts to investigate early computational aerodynamics were carried out and tested but with a restricted understanding of the problem and limited computing power in regards to turbulent flow models and the prediction of separated flows and wake regions. A number of weaknesses in previous automobile body aerodynamic investigations were seen, in particular, the preparation of geometry, generation/adjustment of mesh, turbulence models and assessment of drag coefficient was rarely combined into a holistic process. Investigations of this nature primarily assessed the drag coefficient on both experimental grounds, or via simple estimation, without an indepth analysis of the resulting flow. The drawbacks of traditional aerodynamic drag measurement systems is: high experimentation costs, prototypes are required, a laboratory is required, and they do not offer effective means of refining and assessing designs.

IV. PROPOSED SYSTEM

This system provides a simulation-based framework to decrease aerodynamic drag using computational fluid dynamics (CFD) analysis within ANSYS Fluent. The method is a favorable alternative to traditional experimentation such as wind tunnel testing that requires costly prototypes and excessive time. This system can analyze flow patterns around a passenger car model effectively and efficiently in terms of time and cost to reduce drag coefficient through geometric modification obtained from simulation results. The system combines geometry preparation, meshing, turbulence modeling, solver setup, and post-processing to obtain high prediction accuracy and reduce drag. The crucial areas that contributes to drag can be located by observing pressure distribution, streamlines and wake behind the vehicle and corresponding geometry modifications are performed.

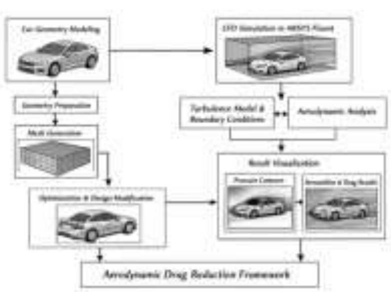

Fig. 1. Architecture of the proposed CFD-based aerodynamic drag reduction framework using ANSYS Fluent.

A. Geometry Preparation and Mesh Generation

This is the first step in the proposed framework where a 3-D model of the vehicle needs to be created and then used as a source for simulation. It needs to be put in the flow domain, simulating practical driving condition. The size of the domain needs to be large enough to avoid boundary influence. Mesh has to be generated using the defined geometry and the generated mesh has to be refined near the surface of the vehicle and in the wake area so that boundary layer effect and flow separation can be modeled accurately. Meshing also affects convergence and accuracy of the drag coefficient value.

B. CFD Solver Configuration and Aerodynamic Analysis

This is the main part of the proposed system where the CFD simulation is performed using ANSYS Fluent. A few boundary conditions like velocity inlet, pressure outlet, and no-slip wall conditions are applied to define the flow characteristics around the car. A turbulence model has to be applied to model complex flow structures like vortices and wake region behind the car. CFD solver gives information on aerodynamics factors such as velocity, pressure distribution, behavior of streamlines and drag coefficient. These parameters can be used to analyze high pressure on the front and low pressure on the rear of the car that contribute to drag.

C. Drag Reduction through Geometry Optimization and Result Visualization

This is the final stage in the proposed framework where modifications on the geometry is made based on the obtained information from CFD simulations, for example, the rounded profile of the rear end, the curvature of the side profile and modifications in the wake region to minimize the separation of the flow. A visualization block has to be implemented which help to determine the aerodynamic performances based on contour plots, streamline, drag coefficients of the model with its optimized geometry. This module helps engineers to analyze and observe flow behaviors for the optimized vehicle. It offers a practical way of modifying the shape of vehicle for efficient use of fuel without using prototype again and again.

V. METHODOLOGY

Reducing aerodynamic drag is a topic of intense interest among present-day automotive engineers, since it plays a significant role in a vehicle's fuel efficiency, its behavior at high speeds, and its stability. The understanding is that aerodynamic effects over a given geometry can be simulated, yielding both a flow map for the car's wake region and a prediction of separation points. Accordingly, ANSYS Fluent software and computational fluid dynamics (CFD) technique were employed in the present study to perform analysis on a passenger car's aerodynamics and determine methods for improving its aerodynamic efficiency.

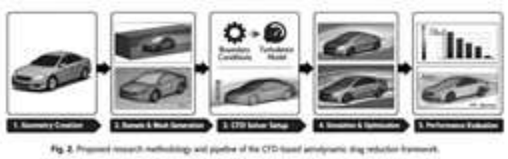

Fig. 2. The CFD based aerodynamic drag reduction system pipeline and the proposed methodology of the research.

The flow chart above displays the pipeline of CFD based system in aerodynamic drag reduction and its application scheme for this project. The project methodology is broken down into 5 stages: creation of geometry, setup of computational domain, meshing and solver setting up and solution of model, analysis and modification of aerodynamic performance, performance evaluation of the model.

1. Geometry Creation

Firstly, 3-D model of passenger car for external aerodynamics is created as the foundation of the simulation. The geometry is appropriately simplified so as to possess sufficient aerodynamic characteristics, but without significant deviation that can alter its aerodynamic behavior. It is required that a careful surface cleaning and preparation is carried out prior to meshing and simulation procedures in order to reduce complications.

2. Meshing and Computational Domain

Next, the prepared geometry of vehicle is placed in a computational flow domain to try to simulate the real environment encountered by the vehicle, which should be sufficiently remote to prevent local flow field of the vehicle to be affected.

Numerical domain then has to be partitioned into finite volume elements, where proper meshing strategy should be applied. Meshes near car's surface and wake should be fine for careful study of boundary layer and separation. Smooth meshing also contributes to numerical accuracy.

3. CFD Solver Setup

Following the modeling of the geometry, the CFD solver, which is ANSYS Fluent in this study, is then used to simulate the flow around the car model as steady state computation. Air is chosen as the working medium, and velocity inlet, pressure outlet, no slip wall condition are set up according to the normal condition of vehicles' operating.

A suitable turbulence model must be employed for the prediction of the detailed flow behavior, including vortices formation, stream-line pattern and wake flow around the car body.

4. Simulation Implementation and Model Optimization

Once the solver is setup, simulation runs until the convergence criterion is satisfied. Behavior such as stream-line pattern, pressure distribution, velocity profile are observed constantly. Parameters like car shape (e.g.Smoothing the rear end, changing curve designs) should be appropriately adjusted to reduce the strength of wake and separation.

5. Performance Evaluation

Finally, various parameters of baseline and optimized models such as drag coefficient, pressure distribution, velocity profiles, and stream-line patterns will be compared, and that helps evaluate the improvement gained from the technique of drag reduction and prove its efficiency.

The result demonstrates that the CFD based model can successfully simulate the aerodynamic characteristics of a vehicle and effectively reduce its drag coefficient without an expenditure of costly physical models. Therefore, car's design can be made more energy-efficient.

VI. CALCULATIONS

The aerodynamic drag force acting on the vehicle is determined using the standard relation between flow properties and body geometry. The drag force can be expressed as:

$$F_d=\frac{1}{2}\rho V^2 C_d A \quad (1)$$

where Fd represents the drag force, ρ is the density of air, V denotes the free-stream velocity, Cd is the drag coefficient, and A corresponds to the projected frontal area of the vehicle.

A. Flow Characterization

To understand the nature of the flow, the Reynolds number is evaluated using:

$$Re=(\rho VL)/\mu \quad (2)$$

Substituting the values:

$$Re=\frac{1.225\times 27\times 4.2}{1.81\times 10^{-5}} \quad (3)$$

$$Re\approx 7.67\times 10^{6} \quad (4)$$

The obtained Reynolds number confirms that the flow regime is turbulent, which justifies the use of turbulence models in the numerical simulation.

B. Baseline Configuration

For the initial vehicle model, the drag coefficient is considered as Cd=0.313C_d = 0.313Cd=0.313. The drag force is computed as:

$$F_d=\frac{1}{2}\times 1.225\times (27)^2\times 0.313\times 2.3 \quad (5)$$

$$= 0.6125 \times 729 \times 0.313 \times 2.3 \quad (6)$$

$$= 446.51 \times 0.313 \times 2.3 \quad (7)$$

$$= 139.77 \times 2.3 \quad (8)$$

$$\approx 321.5\ N \quad (9)$$

Thus, the drag force for the baseline model is approximately 321.5 N.

C. Optimized Configuration

After geometric modifications, the drag coefficient is reduced to Cd=0.285. The corresponding drag force is calculated as:

$$F_d=\frac{1}{2}\times 1.225\times (27)^2\times 0.285\times 2.3 \quad (10)$$

$$= 0.6125 \times 729 \times 0.285 \times 2.3 \quad (11)$$
$$= 446.51 \times 0.285 \times 2.3 \quad (12)$$
$$= 127.25 \times 2.3 \quad (13)$$
$$\approx 292.7\ N \quad (14)$$

Hence, the optimized model experiences a reduced drag force of approximately 292.7 N.

D. Performance Evaluation

The improvement in aerodynamic performance can be quantified in terms of drag reduction efficiency:

$$\eta=\frac{F_{baseline}-F_{optimized}}{F_{baseline}} \quad (15)$$

$$\eta=\frac{321.5-292.7}{321.5} \quad (16)$$
$$= 0.0896 \quad (17)$$
$$= 8.96\% \quad (18)$$

E. Power Requirement Analysis

The power required to overcome aerodynamic drag is calculated using:

$$P=F_d\times V \quad (19)$$

For the baseline model:

$$P=321.5\times 27\approx 8.68kW \quad (20)$$

For the optimized model:

$$P=292.7\times 27\approx 7.90kW \quad (21)$$

The reduction in power requirement indicates Improved aerodynamic efficiency of the optimized design.

VII. RESULTS AND DISCUSSION

CFD simulation of the aerodynamic performance of passenger car model is performed using ANSYS Fluent steady state. Flow pattern around the vehicle surfaces, points of high contribution of aerodynamic resistance are determined by CFD simulation. The parameters such as velocity contours, pressure distribution, streamline patterns and drag coefficient values are used in simulation for both base and modified models of the vehicle.

The baseline model results showed regions of high pressure over the front of the vehicle and low pressure at the back, producing a drag force resisting the motion. The analytical calculation of the Drag Coefficient resulted in the values of 0.313 and Drag Force = 321.5 N for the base model.

To improve the aerodynamic resistance of the car the model geometry was optimized. For this purpose trailing ends were smoothed and body was smoothly curved so that flow doesn't get separated at the back. Thus the flow follows the car body at the rear end.

The simulation was repeated with optimal geometry for a particular set of conditions to observe performance variation. The drag coefficient for modified model was found to be 0.285 andDrag force=292.7 N. Total decrease in drag was 8.96 per cent.

According to velocity contour the turbulence is reduced at wake region of car with respect to the baseline model and pressure distribution over car body was more uniform. From streamline behavior of the model, it was evident that the flow clings to the rear side of car.

This modeled CFD will effectively reduce the aerodynamic resistance of vehicle and helps the designer to test various models instead of physical prototyping in preliminary analysis.

VIII. FUTURE WORK

Although the proposed CFD based aerodynamic drag reduction model provides reasonable and accurate numerical results for the interaction of the airflow over the passenger car model, further improvements may be implemented in subsequent works in order to enhance the simulation accuracy and practicality when applied in a real automotive design environment.

Among these future directions, there will be the implementation of advanced turbulence models, to achieve a more accurate prediction for various structures of wakes and separation flow regions. While currently considering the steady-state flow, subsequent research work may also include a transient flow analysis in order to capture better the time-dependent aerodynamic effects that result in a more representative model of unsteady airflow features and vortex formations that take place over vehicles at various operating conditions in the real world.

The other area that is to be focused in future research is the inclusion of automated geometry optimization techniques. At the moment, the geometric improvements are derived from simulation observations and manual adjustments of the vehicle. However, future studies can combine CFD solvers with optimization algorithms in order to optimize the overall car profile according to aerodynamic efficiency and structural feasibility constraints. Parametric shape optimization and genetic algorithms are some of the optimization tools that may be incorporated for automated geometry optimization of the vehicle profile.

Further possible improvements can also be conducted by incorporating the impact of various aerodynamic features like diffusers, rear spoilers and underbody panels to the simulation model. In this way, their effect on wake reduction and pressure distribution over the car model can be further analyzed so that optimal combination for aerodynamic efficiency can be developed.

Lastly, another way of possible extension of this work is the combination of CFD simulations with experimental studies like wind tunnel testing. Such a hybrid approach could provide more validation of numerical models and a higher confidence on prediction of the performance characteristics.

IX. CONCLUSION

This paper introduces the aerodynamic analysis system, which is a CFD based system, implemented to achieve the research on aerodynamic drag produced by the model of a passenger car, and its drag reduction, using ANSYS Fluent. The presented system can take place of the trial and error to approach the optimal design by way of systematic numerical simulation of the predictive fluid flow characteristics about the structure of automobile. The system quickly locates the aerodynamic losses by visualizing velocity distribution, contour of pressure, streamline distribution and region of wake form, in order to propose the design advises for optimizing the vehicle.

From simulation, results indicates that through improving attaching of the air, reducing the strength of the wake on the rear part of car body, an optimum automobile configuration result in a significant drag reduction comparing to the base configuration. High performance of the pressure distribution and smooth streamline shape indicate that the CFD based optimization algorithms are able to obtain the exterior flow features quite accurately and economically.

Compared to the classical aerodynamic analysis method that primarily depends on the physical wind tunnel test, the proposed system gives a complete chain of geometry preparation, meshing, solver setup, simulation, and post processing within a common environment, so the engineers can examine the several variations of geometries without preparing the real parts. Therefore, it can save significant amount of cost and time for the development.

The structure of the flow field around the car can be easily displayed in the form ofvisualization of aerodynamic parameters(contour plots, stream lines etc.). Understanding the flow can greatly help the designers on understanding the role of different parts of the car on drag generation. Meanwhile, it can achieve quite rapidly on the general workstations by simulation efficiency.

Conclusively, the CFD based aerodynamic drag reduction system offers an engineering application to evaluate aerodynamic efficiencies of the cars before the test. It has been shown that the optimized design will contribute greatly on fuel saving when used an proper system and proper turbulence model.

REFERENCES

1. J. D. Anderson, Computational Fluid Dynamics: The Basics with Applications. New York, NY, USA: McGraw-Hill, 1995.
2. H. Schlichting and K. Gersten, Boundary-Layer Theory, 9th ed. Berlin, Germany: Springer, 2017.
3. S. V. Patankar, Numerical Heat Transfer and Fluid Flow. New York, NY, USA: Hemisphere Publishing, 1980.
4. W. H. Hucho, Aerodynamics of Road Vehicles, 4th ed. Warrendale, PA, USA: SAE International, 1998.
5. J. Katz, Race Car Aerodynamics: Designing for Speed. Cambridge, MA, USA: Bentley Publishers, 2006.
6. F. R. Menter, "Two-equation eddy-viscosity turbulence models for engineering applications," AIAA Journal, vol. 32, no. 8, pp. 1598–1605, 1994.
7. P. R. Spalart and S. R. Allmaras, "A one-equation turbulence model for aerodynamic flows," AIAA Paper 92-0439, 1992.
8. ANSYS Inc., ANSYS Fluent Theory Guide, Release 2023 R1, Canonsburg, PA, USA, 2023.
9. J. Happian-Smith, An Introduction to Modern Vehicle Design. Oxford, U.K.: Butterworth-Heinemann, 2001.
10. S. Pope, Turbulent Flows. Cambridge, U.K.: Cambridge University Press, 2000.
11. L. Davidson, "Fluid mechanics, turbulent flow and turbulence modeling," Chalmers University of Technology, Sweden, Tech. Rep., 2015.
12. M. S. Seddon and S. Goldsmith, Intake Aerodynamics. London, U.K.: Collins Professional & Technical Books, 1999.
13. R. W. Fox, A. T. McDonald, and P. J. Pritchard, Introduction to Fluid Mechanics, 8th ed. Hoboken, NJ, USA: Wiley, 2011.
14. S. Ahmed, G. Ramm, and G. Faltin, "Some salient features of the time-averaged ground vehicle wake," SAE Technical Paper 840300, 1984.
15. T. Cebeci and P. Bradshaw, Physical and Computational Aspects of Convective Heat Transfer. New York, NY, USA: Springer, 1988.