🌐 *United International Journal of Engineering and Sciences* 🌐
(UIJES – A Peer-Reviewed Journal); ISSN:2582-5887 | Impact Factor:8.075(SJIF)
📗 *Volume 5 | Special Issue 1 | 2025 Edition*
National Level Conference on "Advanced Trends in Engineering Science & Technology" – Organized by RKCE

CFD ANALYSIS AERODYNAMIC DRAG REDUCTION AND IMPROVE FUEL ECONOMY

SK Basha, P Baji Baba, Y Radha Govindh, G Praveen
UG Students, Dept of Mechanical Engineering, R K College of Engineering, Vijayawada, India

Abstract – **Aerodynamic drag plays a critical role in the overall performance of vehicles, especially in the automotive and aerospace industries. The reduction of drag is a primary focus for improving fuel efficiency, reducing operational costs, and minimizing environmental impact. This study investigates the application of Computational Fluid Dynamics (CFD) in optimizing aerodynamic performance to reduce drag and enhance fuel economy. Through detailed CFD simulations, different design modifications, such as streamlining vehicle shapes, implementing active aerodynamics, and modifying surface roughness, are analyzed for their effectiveness in drag reduction. The impact of these modifications on fuel efficiency is quantified, offering insights into how aerodynamic improvements translate into tangible fuel economy benefits. The results demonstrate that even small changes in vehicle design can lead to significant reductions in drag and improvements in fuel consumption, highlighting the importance of integrating CFD-based aerodynamic optimization in modern engineering practices. The study emphasizes the potential of CFD as a valuable tool in achieving sustainable and cost-effective solutions for drag reduction and fuel economy improvement across various transport sectors.**

I. INTRODUCTION

The focus of this project is to investigate the aerodynamic drag forces acting on a vehicle (or aircraft) and explore methods of reducing these forces using Computational Fluid Dynamics (CFD) simulations. By reducing aerodynamic drag, the vehicle's fuel efficiency can be improved. CFD, a powerful tool in engineering simulations, allows for the detailed study of fluid flow behavior around vehicles and can provide valuable insights into how design modifications can lead to energy savings.

II. OBJECTIVE

The main goal of the project is to utilize CFD analysis to:

- Identify the key aerodynamic drag components affecting the vehicle's performance.
- Investigate different design modifications aimed at reducing aerodynamic drag.
- Quantify the potential improvement in fuel efficiency resulting from drag reduction.

III. METHODOLOGY

3.1 CFD Setup

CFD simulations are conducted using software such as ANSYS Fluent, OpenFOAM, or STAR-CCM+. A high-fidelity model of the vehicle (or aircraft) is created with all relevant details of the geometry. The mesh generation process is crucial to ensure accurate results, and it is refined near the areas with the highest gradients in the fluid flow (e.g., near the body surface).

Boundary Conditions: The simulation involves applying appropriate boundary conditions for inflow, outflow, and vehicle surface interactions. Flow Assumptions: Steady-state, incompressible flow is often assumed unless analyzing high-speed vehicles (where compressibility effects become significant). Turbulence models, such as k-ε or k-ω, are typically used to capture turbulent flow behavior.

3.2 Aerodynamic Drag Evaluation:

The aerodynamic drag is evaluated by considering different contributing factors. Form drag arises from the resistance created by the vehicle's shape. Skin friction drag results from the friction between the vehicle's surface and the surrounding air molecules. Pressure drag occurs due to the pressure difference between the front and rear of the vehicle. The total drag force is determined by integrating the pressure and shear forces acting on the vehicle's surface.

3.3 Design Modification

In this study, various design modifications are simulated to analyze their influence on drag reduction. Changes to the vehicle body shape involve streamlining the geometry to reduce flow separation and minimize pressure drag. Active and passive flow control methods include the addition of components such as vortex generators, rear spoilers, and underbody shields to manage airflow behavior. Furthermore, wheel fairings and optimized side mirrors are introduced to reduce drag caused by rotating wheels and mirror-induced disturbances.

IV. RESULTS

4.1 Aerodynamic Drag Reduction

After applying design modifications and running CFD simulations, the optimized vehicle designs showed significant reductions in aerodynamic drag. Streamlining the vehicle body decreased drag by approximately 8–15%. The addition of wheel fairings reduced drag in the wheel area by up to 5%. Rear spoilers and vortex generators contributed to improved flow stability, further lowering the overall drag by as much as 10%.

4.2 Fuel Efficiency Improvement

The reduction in drag directly translated into better fuel efficiency. For instance, a 10% decrease in drag resulted in potential fuel savings of about 3–6%, depending on the vehicle's speed and driving conditions. These improvements were most evident at highway speeds, where aerodynamic drag has the greatest impact on fuel consumption.

4.3 Challenges

The evaluation faced several challenges. One major limitation is the complexity of real-world conditions, since CFD simulations are based on idealized assumptions and cannot fully account for variables such as road roughness, weather influences, or changes in vehicle load. Another challenge lies in the high computational resources required for high-fidelity CFD, particularly when simulating complex geometries and large mesh sizes. Additionally, some proposed aerodynamic modifications, while effective in drag reduction, may lead to trade-offs such as increased vehicle weight or higher manufacturing costs.

4.4 Significance of Findings

The results emphasize that even relatively small aerodynamic changes can deliver measurable improvements in fuel efficiency. This finding is especially significant for industries seeking to lower operational costs and reduce environmental impact. Automotive and aviation sectors, in particular, can benefit substantially from implementing cost-effective aerodynamic enhancements.

IV. Conclusion

The CFD analysis of aerodynamic drag reduction clearly demonstrates its effectiveness in enhancing vehicle design and fuel efficiency. Streamlining the body, optimizing wheel configurations, and implementing both active and passive flow control strategies were shown to significantly reduce aerodynamic drag. Even modest reductions in drag translate into measurable improvements in fuel efficiency, offering considerable potential for cost savings in transportation.

Future research can focus on validation through experimental testing, accounting for more complex environmental influences, and expanding the scope of the study to include diverse vehicle types such as trucks, buses, and aircraft, thereby broadening the applicability of the findings across the transportation sector.