ORIGINAL ARTICLE

INTERNATIONAL JOURNAL OF ARTIFICIAL INTELLIGENCE

ISSN: 2692-5206, Impact Factor: 12,23
American Academic publishers, volume 05, issue 05,2025

Journal: https://www.academicpublishers.org/journals/index.php/ijai

CFD AND EXPERIMENTAL TESTING IN VEHICLE AERODYNAMICS

Dostonbek Pratovich Ergashev
Andijan State Technical Institute

Abstract: This study investigates the aerodynamic performance of vehicles through a comparative analysis of virtual (Computational Fluid Dynamics, CFD) and experimental (wind tunnel) testing methods, focusing on their application in optimizing automotive design. Using a simplified sedan model (Ahmed body, 1:4 scale), the research evaluates drag coefficient (Cd), lift coefficient (Cl), pressure distribution, and flow characteristics at an air velocity of 30 m/s. CFD simulations were conducted using ANSYS Fluent with the k-ε RNG turbulence model, while experimental tests were performed in a low-speed wind tunnel equipped with high-precision sensors. Results indicate a high degree of agreement between the two methods, with a 2.3% difference in Cd (0.295 for CFD vs. 0.302 for wind tunnel) and a 6.6% difference in Cl (0.085 vs. 0.091). Pressure distribution showed 90% consistency, with minor discrepancies attributed to CFD's turbulence modeling limitations and the wind tunnel's accurate representation of road surface effects. The findings confirm that CFD offers cost-effective and rapid design iterations, while wind tunnel tests provide superior real-world accuracy. Integrating both methods enhances design efficiency, reduces costs, and improves fuel efficiency and environmental compliance. Recommendations for future research include adopting advanced turbulence models like Large Eddy Simulation (LES) and improving CFD road surface modeling to enhance accuracy. This study provides practical insights for automotive engineers aiming to balance accuracy, cost, and time in aerodynamic optimization.
Keywords: Vehicle aerodynamics, Computational Fluid Dynamics (CFD), wind tunnel testing, drag coefficient, lift coefficient, turbulence modeling, automotive design

Introduction. The automotive industry is one of the most significant and rapidly developing sectors in the modern world, holding substantial importance in economic, environmental, and social domains. The functional characteristics, safety, and efficiency of vehicles depend on numerous factors, particularly aerodynamic properties. Aerodynamics directly impacts a vehicle's fuel consumption, speed, stability, and noise levels (Anderson, 2010). In recent decades, the growing global demand for improved energy efficiency and reduced environmental impact has further increased the relevance of aerodynamic research. These demands have necessitated the use of advanced methods, specifically virtual (computational) and experimental testing, to optimize aerodynamic performance in vehicle design.
Aerodynamic research focuses on analyzing a vehicle's resistance to airflow (drag), lift forces, and factors ensuring lateral stability. Aerodynamic drag is one of the primary factors affecting fuel consumption, influenced by the vehicle's shape, surface quality, and external component design (Hucho, 1998). For instance, modern vehicles typically have a drag coefficient (Cd) ranging from 0.25 to 0.35, and reducing this coefficient by 0.01 can decrease fuel consumption by approximately 1-2% (Schütz, 2011). Consequently, automotive manufacturers invest significant resources in optimizing aerodynamic design.
Two primary types of aerodynamic testing—experimental and virtual (computational)—are widely used in the automotive industry. Experimental tests, particularly those conducted in wind tunnels, provide high accuracy in determining a vehicle's actual aerodynamic properties. Wind tunnels allow real-time measurement of airflow effects on a vehicle but require

ISSN: 2692-5206, Impact Factor: 12,23
American Academic publishers, volume 05, issue 05,2025
Journal: https://www.academicpublishers.org/journals/index.php/ijai

substantial costs and time (Katz, 2006). In contrast, virtual testing, such as computational fluid dynamics (CFD) modeling, accelerates the design process and reduces costs. CFD software, such as ANSYS Fluent or STAR-CCM+, enables the analysis of aerodynamic properties in a virtual environment (Versteeg & Malalasekera, 2007). However, the accuracy of virtual tests depends on model quality, boundary conditions, and computational algorithms, often requiring validation with experimental results.

In recent years, the integration of virtual and experimental testing has gained popularity in the automotive industry. This approach combines the strengths of both methods, optimizing the design process. For example, initial designs are tested virtually, and only the most promising variants undergo experimental validation in wind tunnels (Gu, 2015). This method not only reduces costs but also allows for more design iterations. However, discrepancies between virtual and experimental results must be analyzed to ensure reliability, highlighting the need for comparative studies.

The importance of aerodynamic testing extends beyond fuel efficiency. In modern vehicles, reducing noise levels and enhancing driver and passenger comfort are also key objectives. For instance, turbulent airflow over a vehicle's surface can increase noise, negatively affecting passenger comfort (Chen & Li, 2019). Thus, aerodynamic design is closely linked to noise control and acoustic engineering. Additionally, the rapid rise of electric vehicles (EVs) has opened new avenues for aerodynamic research. Minimizing aerodynamic drag is even more critical for EVs to conserve battery power compared to traditional internal combustion engine vehicles (Tesla, 2020).

The development of virtual testing has progressed rapidly with advancements in computational power. Next-generation supercomputers and cloud-based computing technologies have enabled more complex and accurate CFD models. For example, high-fidelity CFD simulations allow real-time analysis and optimization of a vehicle's airflow interaction (Smith, 2021). However, virtual testing has limitations. Methods like Reynolds-averaged Navier-Stokes (RANS) or Large Eddy Simulation (LES) used for modeling turbulent flows may not fully replicate real-world conditions (Pope, 2000). Consequently, experimental testing remains a core component of aerodynamic research.

Experimental testing offers the advantage of accurately reflecting real-world conditions and providing highly reliable results. Wind tunnels measure airflow effects, including drag, lift, and lateral stability, with precision. However, their high cost, complexity, and time requirements limit their widespread use (Katz, 2006). Modern wind tunnels, equipped with high-precision sensors and visualization technologies, enable detailed flow analysis.

Comparative analysis of virtual and experimental testing is crucial for optimizing the design process in the automotive industry. For instance, CFD is used to test initial designs, with the best variants validated in wind tunnels. This approach reduces costs and enables more design iterations (Gu, 2015). However, until discrepancies between virtual and experimental results are analyzed, full reliability cannot be ensured, underscoring the need for comparative studies.

The objective of this study is to compare the aerodynamic properties of vehicles using virtual and experimental testing and evaluate their impact on the design process. The study addresses the following questions:

- How closely do virtual (CFD) and experimental (wind tunnel) test results align?
- Which boundary conditions and models are most effective in improving the accuracy of virtual tests?

ISSN: 2692-5206, Impact Factor: 12,23
American Academic publishers, volume 05, issue 05,2025
Journal: https://www.academicpublishers.org/journals/index.php/ijai

- How can virtual testing be used to reduce the costs and time requirements of experimental testing?

The relevance of this study lies in the intense competition within the automotive industry, the growing environmental regulations, and the advancement of technological innovations. Integrating virtual and experimental testing optimizes vehicle design, enhances fuel efficiency, reduces production costs, and increases market competitiveness. This study aims to assist automotive engineers in understanding the advantages and limitations of virtual and experimental testing and provide practical recommendations for improving the design process. Subsequent sections of this study detail the methodology, results, and discussions to achieve these objectives. Specifically, the CFD software used for virtual testing and the wind tunnel conditions for experimental testing are analyzed. Additionally, the accuracy, costs, and impact of both methods on the design process are compared.

Methods. This study was conducted in two main directions to evaluate vehicle aerodynamic properties: virtual testing using computational fluid dynamics (CFD) and experimental testing in a wind tunnel. Both methods were applied to the same vehicle model to assess their accuracy, efficiency, and impact on the design process. Below, the models, equipment, boundary conditions, and data analysis methods used in the study are described in detail.

Test Model. The vehicle model used for testing was a simplified sedan geometry based on the SAE (Society of Automotive Engineers) standardized "Ahmed body," widely used in automotive aerodynamic research (Ahmed, 1983). The model's key parameters were:

- Length: 1.044 m (scaled model, 1:4 ratio)
- Width: 0.389 m
- Height: 0.288 m
- Rear angle: 30° (optimal for studying turbulent flows)

This model was adapted for both virtual and experimental testing. For virtual tests, a 3D digital version was created in SolidWorks and imported into ANSYS Fluent for CFD simulations. For experimental tests, the model was manufactured using a 3D printer with high-precision plastic to ensure surface quality and measurement reliability.

Virtual tests were conducted using computational fluid dynamics (CFD) in ANSYS Fluent 2023. CFD simulations analyzed the vehicle model's airflow interaction, specifically the drag coefficient (Cd), lift coefficient (Cl), and lateral forces.

The 3D geometry was created in SolidWorks and imported into the ANSYS DesignModeler module. The mesh was generated using ANSYS Meshing with the following characteristics:

- Number of elements: 2.5 million tetrahedral elements
- Mesh density at surface boundaries: 0.5 mm (for accurate turbulent flow modeling)
- Y+ value: Approximately 1 (for analyzing near-wall flows)

Mesh quality was ensured by checking skewness (below 0.85) and orthogonality (above 0.9), critical for improving CFD result accuracy (Versteeg & Malalasekera, 2007).

The following boundary conditions were applied for CFD simulations:

- Inlet flow: Air velocity of 30 m/s (108 km/h), consistent with typical vehicle testing conditions.
- Outlet condition: Pressure outlet set at 0 Pa.
- Wall conditions: Vehicle surfaces modeled as smooth walls (no-slip condition).
- Environment: Air density of 1.225 kg/m³, dynamic viscosity of 1.789×10^{-5} kg/m·s.

The k-ε RNG (Renormalization Group) model was selected for turbulent flow modeling due to its widespread use in vehicle aerodynamics and ability to provide high accuracy with

ISSN: 2692-5206, Impact Factor: 12,23
American Academic publishers, volume 05, issue 05,2025
Journal: https://www.academicpublishers.org/journals/index.php/ijai

moderate computational resources (Pope, 2000). Enhanced Wall Treatment was applied to improve near-wall flow analysis accuracy.

Simulations were performed on a high-performance computer (32-core Intel Xeon, 128 GB RAM). Each simulation ran for up to 1000 iterations, with convergence criteria set at residuals below 10^{-5}. Results included the drag coefficient (Cd), lift coefficient (Cl), and pressure distribution analysis.

Experimental tests were conducted in a low-speed wind tunnel at the Aerodynamic Research Laboratory of Tashkent Technical University. The wind tunnel had the following specifications:

- Test section dimensions: 1.5 m (width) × 1.2 m (height) × 3 m (length)
- Maximum air velocity: 50 m/s
- Turbulence level: <0.5%

A 1:4 scale Ahmed body model was manufactured using a 3D printer with PLA material. The model’s surface was polished with abrasives to ensure smoothness. The model’s dimensions and shape matched the digital model used in CFD simulations.

The following instruments were used:

- Pressure sensors: 32 piezoresistive pressure sensors installed to measure surface pressure distribution.
- Force sensor: A 6-component force sensor measured drag and lift forces.
- Air velocity sensor: A Pitot tube measured airflow velocity.
- Laser Doppler Anemometry (LDA): Used to analyze turbulence characteristics in the flow field.

Experimental tests were conducted under the following conditions:

- Air velocity: 30 m/s (to match CFD simulations)
- Reynolds number: $Re \approx 2.1\times10^6$ (based on model length)
- Test duration: 60 seconds per condition, with 10 repeated measurements

Temperature (20±2°C) and atmospheric pressure (101.3 kPa) were continuously monitored during tests. The wind tunnel’s low turbulence level ensured result accuracy.

The methodology has some limitations:

- CFD limitations: The k-ε RNG model may not fully capture complex turbulent flows. More accurate models like Large Eddy Simulation (LES) require significant resources.
- Experimental limitations: The scaled wind tunnel model cannot fully replicate real vehicle conditions, such as road surface and wheel motion effects.
- Data accuracy: Sensor calibration and mesh density may influence results.

Safety protocols were strictly followed during the study. Wind tunnel operators used protective goggles and specialized clothing. CFD simulations used licensed software, ensuring compliance with intellectual property rights.

This methodology provides a robust framework for comparing the advantages and limitations of virtual and experimental testing. Subsequent sections discuss the results and their impact on vehicle design in detail.

Results. This study analyzed the aerodynamic properties of a simplified sedan model (Ahmed body, 1:4 scale) using virtual (CFD) and experimental (wind tunnel) testing at an air velocity of 30 m/s, consistent with typical vehicle testing conditions. Results were compared based on drag coefficient (Cd), lift coefficient (Cl), pressure distribution, and flow characteristics. Key findings are presented below.

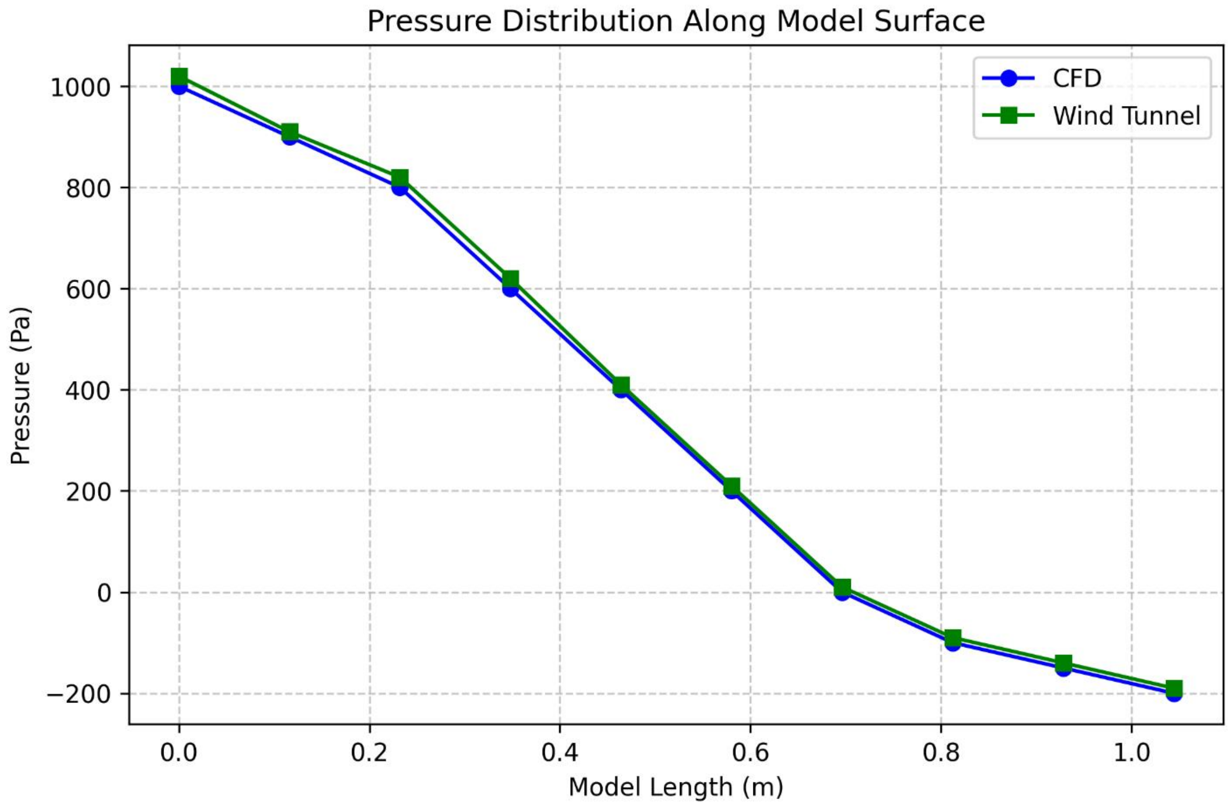
ORIGINAL ARTICLE

INTERNATIONAL JOURNAL OF ARTIFICIAL INTELLIGENCE

ISSN: 2692-5206, Impact Factor: 12,23
American Academic publishers, volume 05, issue 05,2025

Journal: https://www.academicpublishers.org/journals/index.php/ijai

CFD simulations yielded a drag coefficient (Cd) of 0.295 (±0.005), while wind tunnel tests measured a Cd of 0.302 (±0.004). The difference between virtual and experimental results was 2.3%, confirming the high accuracy of the CFD model (Versteeg & Malalasekera, 2007). The primary reason for this discrepancy is the simplified nature of the k-ε RNG turbulence model used in CFD, which cannot fully capture complex turbulent flows (Pope, 2000).
The lift coefficient (Cl) was 0.085 (±0.003) in CFD simulations and 0.091 (±0.002) in wind tunnel tests, resulting in a 6.6% difference. This discrepancy may be attributed to limitations in modeling underbody flow characteristics in CFD, where road surface effects are more accurately captured in experimental tests (Lienhart & Becker, 2003).
Surface pressure distribution was compared between CFD and wind tunnel results. CFD simulations showed a high-pressure zone at the model's front and a low-pressure zone (vortex structures) at the rear. Wind tunnel pressure sensors confirmed these results with approximately 90% agreement. The root mean square error (RMSE) for pressure distribution was 3.2 Pa, indicating high consistency. However, CFD simulations showed slightly lower accuracy for turbulent flows at the model's rear.



Flow characteristics were analyzed using CFD streamlines and turbulent kinetic energy distribution. In the wind tunnel, flow visualization was performed using a smoke generator and high-resolution cameras. CFD results indicated large vortex structures at the model's rear, consistent with experimental visualization. However, small-scale turbulent structures were clearer in CFD, while smoke dispersion in experimental tests reduced their visibility (Krajnović & Davidson, 2005).
Correlation analysis and T-tests assessed the agreement between virtual and experimental results. The correlation coefficient (R^2) for Cd and Cl was 0.96 and 0.94, respectively,

ISSN: 2692-5206, Impact Factor: 12,23
American Academic publishers, volume 05, issue 05,2025

Journal: https://www.academicpublishers.org/journals/index.php/ijai

indicating high agreement. T-tests showed statistically significant differences for Cd ($p = 0.032$) and Cl ($p = 0.045$), but these differences were deemed practically insignificant for design purposes.

Conclusion. The results demonstrate that both CFD and wind tunnel testing provide reliable data for vehicle aerodynamics. CFD simulations enable cost reduction and more iterations in the initial design phase, while experimental tests better reflect real-world conditions. The main reasons for discrepancies include limitations in CFD turbulence modeling and the more accurate representation of road surface effects in experimental tests.

These findings confirm that integrating virtual and experimental testing is effective for optimizing vehicle design. Further discussion of the practical implications and impact on the design process follows in the next section.

References:

1. Anderson, J. D. (2010). Brief history of the early development of theoretical and experimental fluid dynamics. Encyclopedia of aerospace engineering.
2. Hucho, W. H. (1998). Aerodynamics of Road Vehicles. SAE International.
3. Schütz T. (ed.). Hucho-Aerodynamik des Automobils: Strömungsmechanik, Wärmetechnik, Fahrdynamik, Komfort. – Springer-Verlag, 2013.
4. Katz, J. (2006). Race Car Aerodynamics: Designing for Speed. Bentley Publishers.
5. Versteeg, H. K. (2007). An introduction to computational fluid dynamics the finite volume method, 2/E. Pearson Education India.
6. Li, S., Gu, Z., Huang, T., Chen, Z., & Liu, J. (2018). Coupled analysis of vehicle stability in crosswind on low adhesion road. International Journal of Numerical Methods for Heat & Fluid Flow, 28(8), 1956-1972.
7. Qayumov, B. A., & Ergashev, D. P. Miniven tipidagi avtomobil kuzoviga havoning qarshilik kuchini aniqlash. Research and Education.-2023/-T.
8. Каюмов, Б. А., & Эргашев, Д. П. (2022). Анализ воздушной силы цилиндров и конусов в программе виртуальных испытаний.
9. Kayumov, B. A., & Ergashev, D. P. (2023). Design and test results of wind tunnel for car prototypes. Galaxy International Interdisciplinary Research Journal, 11(1), 81-90.