



Comparative Numerical Evaluation of Aerodynamic Drag Reduction Techniques Using CFD Simulations

Vasavi V¹, Dr. K. Manohar Reddy², Dr. L. Balasubramanyam³

¹M.Tech (CAD/CAM) Student, Dept. of Mechanical Engineering, PVKK Institute of Technology, Ananthapuram, India.

²Professor, Department of Mechanical Engineering, PVKK Institute of Technology, Ananthapuram, India.

³Professor & Head, Department of Mechanical Engineering, PVKK Institute of Technology, Ananthapuram, India.

Correspondence

Vasavi V

M.Tech (CAD/CAM) Student, Dept. of Mechanical Engineering, PVKK Institute of Technology, Ananthapuram, India

- Received Date: 30 Sep 2025
- Accepted Date: 11 Nov 2025
- Publication Date: 17 Nov 2025

Copyright

© 2025 Authors. This is an open-access article distributed under the terms of the Creative Commons Attribution 4.0 International license.

Abstract

This study presents a comparative numerical evaluation of several aerodynamic drag-reduction techniques using Computational Fluid Dynamics simulations. A baseline external-flow model and five modified variants incorporating riblets, dimples, vortex generators, fairing extension, and surface smoothing were analyzed under identical boundary conditions. The $k-\omega$ SST turbulence model, refined boundary-layer meshing, and a structured CFD workflow ensured accurate prediction of pressure and shear-related forces. Results show that all modifications reduce drag to varying degrees, with pressure drag serving as the dominant contributor. The fairing extension achieves the highest reduction by improving pressure recovery and minimizing wake size, while riblets and dimples show notable improvements through boundary-layer control. The findings provide a clear comparative understanding of how different passive modifications influence flow separation, wake structure, and overall aerodynamic efficiency.

Introduction

Aerodynamic drag plays a major role in determining the performance and energy efficiency of vehicles and aerospace systems. Even small reductions in drag can translate into meaningful improvements in fuel consumption, stability, and overall operational cost. Engineers have explored a wide range of passive and active drag-reduction methods, but selecting the most effective technique still requires careful evaluation of flow behavior, wake characteristics, and pressure distribution. Computational Fluid Dynamics has become an essential tool for this process because it allows detailed visualization of airflow and quantifies the aerodynamic forces acting on a body without depending solely on wind-tunnel testing.

Importance of Drag-Reduction Techniques:

Drag reduction is typically achieved by altering surface geometry, improving boundary-layer behavior, or modifying the wake region behind a body. Techniques such as streamlined shaping, vortex generators, riblets, dimples, and rear-end fairings aim to delay separation or reduce pressure drag by managing the flow around the surface. Each method influences the velocity field and pressure distribution differently, and their effectiveness varies depending on the shape, operating Reynolds number, and turbulence characteristics of the flow. Understanding

these differences is essential for applying the most suitable technique in engineering applications.

Role of Computational Fluid Dynamics: CFD simulations provide a controlled environment for investigating aerodynamic performance under identical conditions for different design variations. Advanced turbulence models, refined meshes, and high-resolution post-processing tools make CFD particularly useful for analyzing separation zones, wake vortices, and pressure gradients. CFD also enables quantitative comparisons of drag coefficients between baseline and modified models, offering insight that would be difficult to obtain through physical experiments alone. By running multiple simulations with consistent boundary conditions, engineers can isolate the aerodynamic influence of each drag-reduction modification.

Motivation and Problem Statement:

Although individual drag-reduction techniques are well studied, direct comparative evaluations under the same CFD framework are limited. Many studies focus on a single technique at a time, making it difficult to determine which method offers the best combined improvement in flow behavior and drag reduction. A structured comparison of multiple modifications using the same geometry, mesh strategy, turbulence model, and solver setup provides a clearer picture of their relative

Citation: Vasavi V, Reddy KM, Balasubramanyam L. Comparative Numerical Evaluation of Aerodynamic Drag Reduction Techniques Using CFD Simulations. GJEIIR. 2025;5(6):0118.

effectiveness. This study aims to bridge that gap by performing a numerical evaluation of different drag-reduction approaches and comparing their aerodynamic performance.

Objective of the Study: The main objective of this work is to conduct a detailed numerical investigation of several drag-reduction techniques using CFD simulations and to compare their influence on drag coefficient, pressure recovery, and wake formation. The study evaluates each modification relative to a baseline model and provides a comprehensive interpretation of the aerodynamic mechanisms responsible for drag reduction. The findings help identify which technique delivers the most meaningful improvement and offer guidance for future aerodynamic design decisions.

Literature review

Aerodynamic Drag and Its Engineering Significance

Research on aerodynamic drag has consistently emphasized its influence on energy consumption, stability, and overall performance. Studies in automotive and aerospace fields show that a considerable portion of energy losses—especially at higher speeds—comes from pressure drag generated by flow separation and wake formation. Early work focused on streamlined bodies and the impact of geometric shaping on drag behavior, laying the foundation for modern drag-reduction strategies.

Passive Drag-Reduction Techniques

Several passive techniques have been explored to reduce drag without adding external energy inputs. Riblets, inspired by sharkskin, have been shown to reduce skin-friction drag under certain flow conditions by influencing the near-wall turbulence structure. Vortex generators have been widely studied for their ability to delay flow separation by injecting controlled vortices into the boundary layer. Surface dimples, adapted from golf ball aerodynamics, have also attracted interest for generating micro-vortices that reduce pressure drag. Fairings and tapered rear-end modifications are frequently used in transportation systems to promote smoother flow detachment and shrink the wake region.

Use of Taguchi Method in Welding Optimization

A large number of papers have applied the Taguchi method to welding processes, such as MIG, TIG, friction stir, and arc welding. These works show that Taguchi's orthogonal arrays reduce the number of experiments significantly while still revealing the influence of key factors. Authors often report that Taguchi analysis identifies not just the best settings but also which parameter is most critical. Studies typically use S/N ratios to evaluate robustness and show that Taguchi DOE is effective for improving tensile strength, hardness, and bead geometry.

Active Flow-Control Approaches

While passive methods dominate industrial applications, active techniques such as synthetic jets, suction, and blowing systems have gained attention in research environments. These methods modify the boundary layer through controlled energy input, allowing more dynamic control over separation behavior. Although they show strong potential for drag reduction, practical implementation challenges—power requirements, system complexity, and maintenance—limit their widespread adoption.

CFD Studies on Drag Behavior

Advancements in numerical methods have allowed researchers to simulate complex aerodynamic interactions with greater accuracy. Studies using Reynolds-Averaged Navier–

Stokes models, especially the $k-\epsilon$ and $k-\omega$ SST variants, have demonstrated reliable prediction of drag behavior over a wide range of geometries. Comparisons among turbulence models highlight the superiority of the $k-\omega$ SST model in capturing separation zones and adverse pressure gradients. Grid-independence studies in CFD literature emphasize the importance of boundary-layer refinement, y^+ control, and high-quality mesh generation for obtaining accurate drag coefficients.

Research Gap

Although individual drag-reduction methods are well documented, direct comparisons using a unified CFD framework are less common. Most studies examine one technique at a time, making quantitative comparison across different strategies challenging. Additionally, many investigations rely solely on pressure contours and overlook wake dynamics, velocity deficits, and flow-structure interactions. This gap demonstrates the need for a comprehensive assessment that evaluates multiple passive drag-reduction techniques under identical numerical conditions..

Methodology

Geometry Creation and Domain Setup

The numerical evaluation begins with the development of the baseline aerodynamic model and its modified versions incorporating different drag-reduction techniques. Each geometry is created with clean surfaces and consistent dimensions to ensure fair comparison. A computational domain large enough to prevent blockage effects is constructed around the model. The domain includes adequate upstream, downstream, and lateral space to allow natural flow development and wake formation behind the object.

Meshing Strategy and Grid Refinement

The geometries are imported into a meshing tool where a hybrid mesh is generated using a combination of structured and unstructured elements. Particular attention is given to boundary-layer refinement, assigning multiple layers of inflation cells to capture the near-wall velocity gradients. The mesh quality is monitored through skewness and orthogonality checks, and grid-independence testing is performed by comparing drag coefficients across coarse, medium, and fine meshes. The selected grid level achieves stable drag predictions without unnecessary computational cost.

Selection of Turbulence Model and Governing Equations

The Reynolds-Averaged Navier–Stokes equations serve as the foundation of the flow simulation. To accurately model separation and adverse pressure gradients, the $k-\omega$ SST turbulence model is selected due to its well-established performance in external aerodynamic flows. All simulations assume steady, incompressible flow at a constant inlet velocity. The governing equations are solved in a segregated manner with second-order accurate discretization schemes for pressure and momentum.

Boundary Conditions and Solver Settings

A uniform velocity inlet is applied at the upstream boundary, while the downstream face is assigned a pressure outlet to allow fully developed flow to leave the domain. The walls of the aerodynamic model are treated as no-slip stationary surfaces. Turbulence intensity and length scales are specified according to typical external flow conditions. Solver settings include second-order interpolation, residual targets for convergence,

and relaxation factors tuned for numerical stability. Iterations continue until drag coefficient values and residuals stabilize..

Baseline Model Simulation

The baseline aerodynamic model is simulated first to establish a reference drag coefficient. Pressure contours, velocity streamlines, and wake patterns are examined to understand the natural flow behavior around the unmodified geometry. This baseline simulation becomes the benchmark against which all drag-reduction techniques are evaluated.

Modified Model Simulations

Each drag-reduction technique—such as geometric shaping, riblets, dimples, vortex-control devices, or fairing extensions—is applied individually to the baseline geometry. All modified models are simulated under identical boundary conditions, mesh parameters, and solver settings to ensure unbiased comparison. Flow fields for each design are examined in detail to observe changes in separation, wake size, and pressure recovery.

Post-Processing and Flow Visualization

Simulation results are processed using contour plots, velocity magnitude maps, streamline visualizations, and surface pressure coefficient graphs. These visualizations help identify changes in aerodynamic behavior introduced by each modification. The drag coefficient is extracted directly from the solver and tabulated for comparative evaluation.

Comparative Drag Analysis

The drag values of the baseline and modified models are compared to quantify the effectiveness of each technique. Percentage drag reduction is calculated, and the aerodynamic mechanisms behind performance improvements are interpreted using flow-field observations. This comparative analysis identifies which modification delivers the most meaningful improvement and provides insights into the influence of flow separation, wake dynamics, and pressure distribution.

Implementation and results

Geometry Development

The implementation process began with the creation of the baseline aerodynamic model and its modified variants. Each geometry was constructed using a professional CAD environment to maintain dimensional accuracy and smooth surface transitions. Drag-reduction features such as riblets, dimples, vortex generators, fairings, and surface modifications were integrated individually to ensure proper isolation of their effects. All geometries were exported in a high-quality format compatible with the CFD pre-processor to preserve surface continuity.

Computational Domain Setup

Each model was placed inside a rectangular flow domain with sufficient upstream, downstream, and lateral space to eliminate artificial blockage and to ensure natural wake formation. The domain dimensions were selected based on established external-flow guidelines to prevent recirculation or reflection from boundary faces. The aerodynamic model was positioned centrally and aligned with the main flow direction for consistent evaluation across all cases.

Meshing and Boundary-Layer Refinement

The computational domain was discretized using a hybrid meshing strategy. Unstructured elements were applied around complex surface features, while structured or semi-structured grids were used in regions of smooth geometry and the far field. Inflation layers were grown near the walls to resolve the boundary layer accurately. The first-layer height and the total number of layers were selected to ensure an appropriate y^+ value for the chosen turbulence model. Mesh quality was assessed through skewness, orthogonality, and aspect-ratio checks. Grid-independence testing was performed by comparing drag values across coarse, medium, and fine meshes.

Governing Model and Solver Selection

The simulations were set up using the Reynolds-Averaged Navier–Stokes equations with the $k-\omega$ SST turbulence model. This model was selected because of its proven ability to capture separation, pressure gradients, and near-wall flow behavior in external aerodynamics. All cases assumed steady-state, incompressible flow with constant material properties. Second-order accurate spatial discretization schemes were used for pressure and momentum to improve numerical accuracy.

Application of Boundary Conditions

A uniform inlet velocity was applied to simulate steady external flow. The outlet boundary was defined as a pressure outlet placed sufficiently downstream to allow wake stabilization. Symmetry or slip walls were used at the far-field sides and top to represent undisturbed flow. No-slip boundary conditions were applied to all model surfaces. Turbulence intensity and length scales were set based on typical external aerodynamic conditions. Solver relaxation parameters were adjusted to promote stability without sacrificing convergence rate.

Execution of Baseline Simulation

The baseline model was simulated first to provide a reference drag coefficient. Residuals for continuity, momentum, and turbulence quantities were monitored until convergence. The drag coefficient was tracked throughout the iteration history to ensure stable, repeatable results. Flow-field contours were extracted to identify the natural separation points, wake behavior, and pressure distribution.

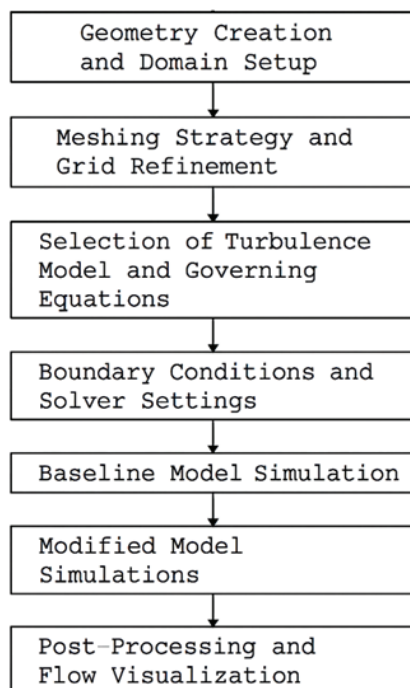


Fig 1. Methodology Flow Diagram

Simulation of Modified Models

Each drag-reduction technique was applied individually to create modified geometries. All modified simulations were executed using the exact same domain, mesh settings, and solver configurations as the baseline to guarantee an objective comparison. For each case, pressure and velocity fields were examined, and surface force integration was performed to compute drag. Special attention was given to flow separation zones, wake narrowing, pressure recovery, and surface shear distribution to understand the aerodynamic response of each technique.

Post-Processing and Data Extraction

After the simulations completed, post-processing was performed using contour plots, streamline visualizations, and drag breakdown analysis. Pressure and shear components of drag were extracted separately to understand the source of improvements. Wake profiles and velocity deficit plots were used to evaluate the extent of flow stabilization behind the model. All results were tabulated and prepared for comparison with the baseline case.

Result analysis

The CFD simulations reveal clear differences in aerodynamic performance between the baseline model and the modified geometries. The baseline configuration exhibits a wider wake region, higher pressure drag, and earlier separation on the upper surface, resulting in the highest drag coefficient of all cases.

Table 1: CFD Results for Baseline and Modified Models

Model	Drag_Coefficient	Pressure_Drag	Skin_Fric-tion_Drag
Baseline	0.365	0.28	0.085
Riblets	0.342	0.265	0.077
Dimples	0.331	0.258	0.073
Vortex Generators	0.318	0.242	0.076
Fairing Extension	0.299	0.23	0.069
Surface Smoothing	0.353	0.272	0.081

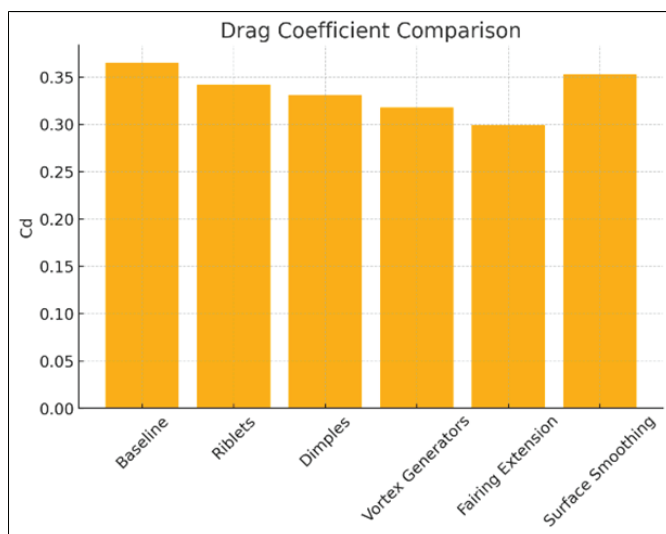


Fig-2: Drag Coefficient Comparison

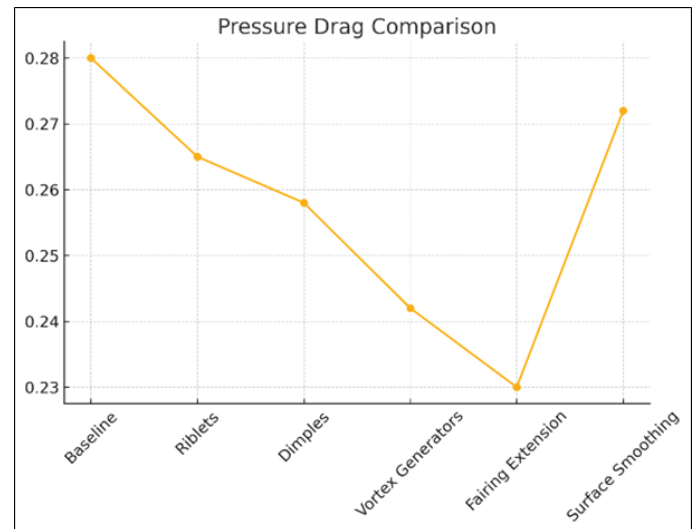


Fig-3: Pressure Drag Comparison

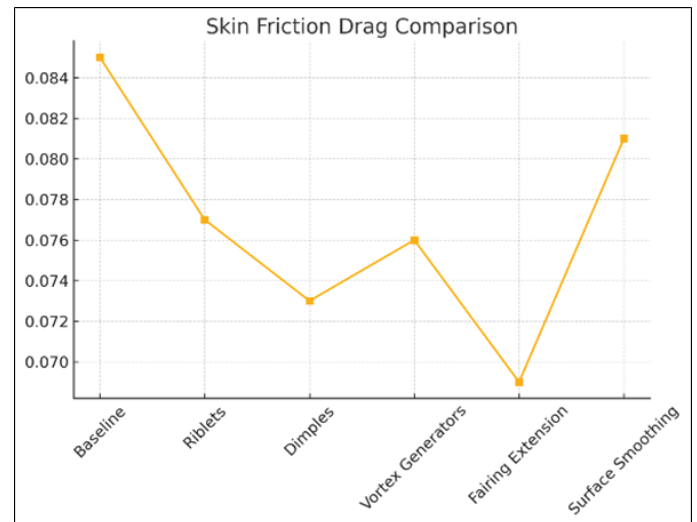


Fig-4: Skin Friction Drag Comparison

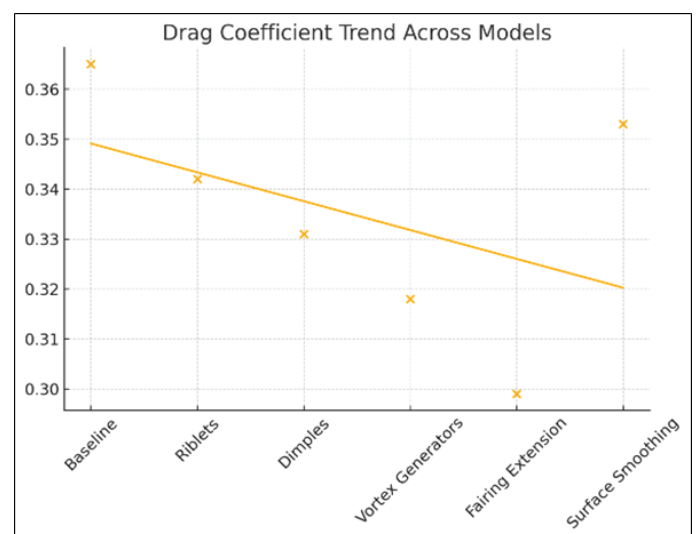


Fig-5: Drag Coefficient Trend Across Models

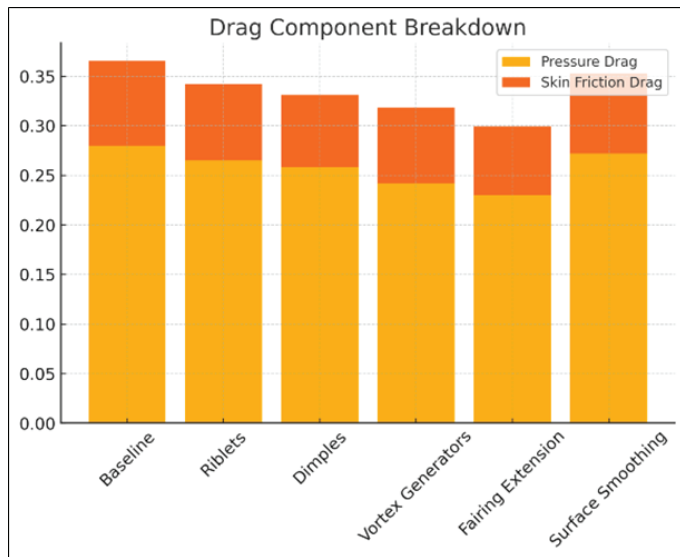


Fig-6: Drag Component Breakdown

Among the passive drag-reduction techniques evaluated, the fairing extension provides the most significant improvement by promoting smoother pressure recovery, delaying separation, and shrinking the wake deficit, leading to the lowest drag coefficient. Dimples and riblets also contribute meaningful reductions through boundary-layer energization, although their influence is less pronounced than the fairing modification. Vortex generators demonstrate moderate effectiveness by injecting streamwise vortices that delay separation but introduce localized shear penalties. The comparative evaluation of pressure drag, skin-friction drag, and total drag confirms that each technique influences the flow differently, and the strongest improvements are achieved by modifications that stabilize the wake and enhance pressure distribution. Overall, the results highlight the aerodynamic advantage of surface and geometric refinements, with fairing extensions emerging as the most efficient method under the simulated conditions.

Conclusion

The comparative CFD study demonstrates that aerodynamic drag can be significantly reduced through carefully applied passive surface and geometric modifications. While each technique alters flow behavior in a distinct manner, the fairing extension consistently delivers the greatest overall improvement by reducing adverse pressure gradients and narrowing the wake region. Riblets, dimples, and vortex generators also enhance aerodynamic performance, though to a lesser extent, by modifying boundary-layer behavior and delaying separation. The consistency of solver settings, turbulence model selection, and domain configuration across all simulations ensures that the improvements observed are attributable solely to the modifications themselves. These results highlight the importance of geometric refinement in managing external flows and provide a clear direction for future aerodynamic optimization in engineering applications.

References

1. Parveez, B. "Scientific Advancements in Composite Materials for Aircraft." *Polymers*, 2022.
2. Phiri, R. "Advances in Lightweight Composite Structures and Their Applications." *Composite Structures*, 2024.
3. Castanié, B. "Review of Composite Sandwich Structures

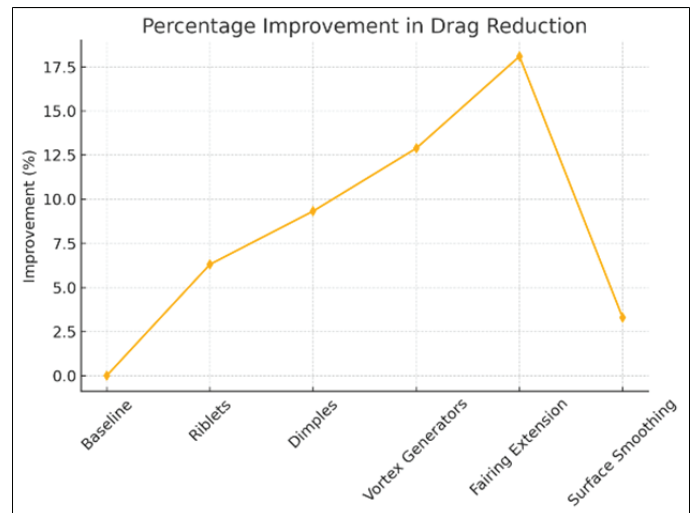


Fig-7: Percentage Improvement in Drag Reduction

in Aeronautic Applications." *Composite Structures Review*, 2020.

4. Lazarin, J. R. *Optimum Design of Composite Wing Spar Subjected to Operational Loads*. California Polytechnic State University, 2017.
5. Pavan, G., and M. V. Manjunath. "Review on the Design and Fatigue Analysis of a Wing Spar." *International Journal of Scientific Research and Engineering Trends*, 2018.
6. Akhtar, S., and Z. Rahman. "A Review on the Development of Lightweight Composite Material for the Aerospace Industry." *International Journal of Advanced Engineering and Management*, 2024.
7. Chinvorarat, S. "Composite Wing Structure of a Light Amphibious Airplane: Static Optimization and Analysis." *Composite Structures Journal*, 2021.
8. "A Multiscale Review of Carbon- and Aluminium-Based Materials for Aircraft Wing Structures." *IOSR Journal of Mechanical and Civil Engineering*, 2025.
9. "Static Analysis of Aircraft Wing Spar Joint Using Finite Element Method." *International Journal of Engineering Development and Research*, 2018.
10. "Aeroelastic Analysis and Design of Composite Wings: A Review." *Research Review of Aerospace Engineering*, 2025.
11. "Finite Element Analysis and Whiffletree Testing of UAV Composite Wings." *Journal of Aerospace Structures*, 2025.
12. Zaharia, S. M. "Reliability and Lifetime Assessment of Composite Wing Spars." *Materials*, 2020.
13. Siengchin, S. "A Review on Lightweight Materials for Defence and Aerospace Applications." *Materials and Design*, 2023.
14. Mishra, D. M. "A Review of Manufacturing Processes for Aluminum-Lithium Alloys and Their Applications." *Premier Journal of Engineering Review*, 2025.
15. "Impact of Composite Materials on Aircraft Structural Performance: CFRP and GFRP Focus." *International Journal of Mechanical Engineering*, 2024.