International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Finite Element Analysis of Wing Rib Structure under Aerodynamic and Structural Loads

T J Prasanna Kumar[*1], S. Yogendra [2a], Sk. Meera Shareef [2b], Y.Rakesh [2c], V. Kalyan Sai Ram [2d], V. Bhanu Nagendra [2e]*

[1*] Assistant Professor, Department of Mechanical Engineering, PVPSIT, Vijayawada-520007, India
[2a,b,c,d,e] UG Students, Department of Mechanical Engineering, PVPSIT, Vijayawada-520007, India
Email: tjpk@pvpsiddhartha.ac.in

ABSTRACT

This study conducts a Finite Element Analysis (FEA) on an aircraft wing rib to assess its structural behavior under aerodynamic and mechanical loads. The wing rib, a crucial component in an aircraft's wing assembly, provides support to the wing skin and helps maintain the aerodynamic shape while distributing loads efficiently.

Using ANSYS 13.0, the rib was modeled with precise geometric features, including lightening holes, and meshed with appropriate finite elements to ensure computational accuracy. The analysis was performed under cantilever beam assumptions, meaning that one end of the rib was fully constrained while various loads, including aerodynamic pressure and inertial forces, were applied to simulate real-world conditions.

The primary objective of this study is to evaluate the stress distribution and deformation patterns under these applied loads, ensuring that the rib structure can withstand operational stresses without failure. By identifying regions of high stress concentration, potential failure points can be determined, leading to recommendations for design optimizations such as material reinforcements, modifications to the cutout geometry, or alternative material selection.

The results of the analysis reveal critical stress zones near attachment points and cutouts, emphasizing the need for localized strengthening. Additionally, deformation analysis provides insights into the overall flexibility and stability of the rib, guiding improvements for enhanced structural integrity and weight efficiency in future designs.

1. Introduction

Aircraft wing ribs play a crucial role in maintaining structural integrity and load distribution. Accurate stress analysis is vital to prevent structural failure and ensure weight optimization. This study employs FEA to simulate stress distribution and deformation in a rib structure under representative loading conditions.

1.1 Importance of Stress Analysis

The accurate assessment of stress and deformation in wing ribs is critical for several reasons:

Preventing Structural Failure: Overloading or fatigue-induced stress can lead to cracks and structural failure. Proper analysis helps in identifying high-stress regions.

Weight Optimization: Aircraft design emphasizes lightweight materials and efficient geometries to enhance fuel efficiency and performance. Optimizing rib structures reduces unnecessary weight without compromising strength.

Improving Durability and Safety: By understanding stress distributions and deformation patterns, engineers can enhance the longevity and safety of wing components.

Bruhn, E. F [1], author presented fundamental resource for aircraft structural analysis, covering stress distribution, load transfer, and failure criteria. Megson, T. H. [2], provides an in-depth discussion on wing rib design, load distribution, and stress analysis using modern computational techniques. Rao, S. S [3], presents a comprehensive reference on FEA, including applications in aerospace structures such as wing ribs and spars. Niu, M. C. Y [4], the author vovers practical aspects of aircraft structural design, including the role of wing ribs and stress analysis techniques. Reddy, J. N [5], author explains FEA concepts applied to aerospace engineering, including stress analysis of wing ribs. Mansouri, H., & Lakis, A. A [6], he Discusses the application of FEA in aircraft wing components, including ribs, for structural and vibrational analysis.

2. Methodology

2.1 Overview of Finite Element Analysis (FEA)

Finite Element Analysis (FEA) is a numerical method used to analyze and predict the behavior of complex engineering structures under various loading conditions. It works by discretizing a structure into a finite number of smaller elements, allowing for the computation of stress, strain, displacement, and deformation at each element. This approach is especially useful for aerospace applications, where structural components like wing ribs must withstand aerodynamic, inertial, and operational loads.

2.1.1 Application of FEA in Wing Rib Structural Analysis

In this study, FEA was employed to perform a **structural evaluation** of an aircraft wing rib under realistic flight conditions. The primary objectives were to:

1. Determine Stress Distribution: Identify regions of high stress concentration that may lead to failure or fatigue.
2. Analyze Deformation Characteristics: Evaluate the displacement of the rib under applied loads to ensure it remains within allowable limits.
3. Optimize Structural Design: Assess the influence of rib geometry, material properties, and loading conditions to suggest weight-efficient design improvements.

2.1.2 Modeling and Meshing

The wing rib was modeled using ANSYS 13.0, a widely used FEA software for aerospace applications. The rib geometry was developed based on realistic aircraft specifications, incorporating cutouts for weight reduction. The meshing process involved:

- Discretization of the Rib Structure: The rib was divided into small finite elements to enable accurate calculations.
- Mesh Refinement at Critical Locations: Areas around cutouts and load-bearing sections were assigned a finer mesh to capture stress variations accurately.

2.1.3 Material Properties

The analysis considers Aluminum 2024-T3, a common aerospace material, with properties as follows:

Young's Modulus (E)	73.1 GPa
Poisson's Ratio (ν)	0.33
Density (ρ)	2770 kg/m³
Yield Strength, (σ_{yield})	345 MPa

2.2 Geometry and Mesh

A 3D model of the wing rib was created, featuring three lightening holes to reduce weight while maintaining strength. The structure was meshed using quadrilateral elements:

Length:	900 mm
Height:	300 mm
Thickness:	3 mm
Mesh Type:	Quadrilateral elements
Element Size:	5 mm
Total Elements:	~10,000

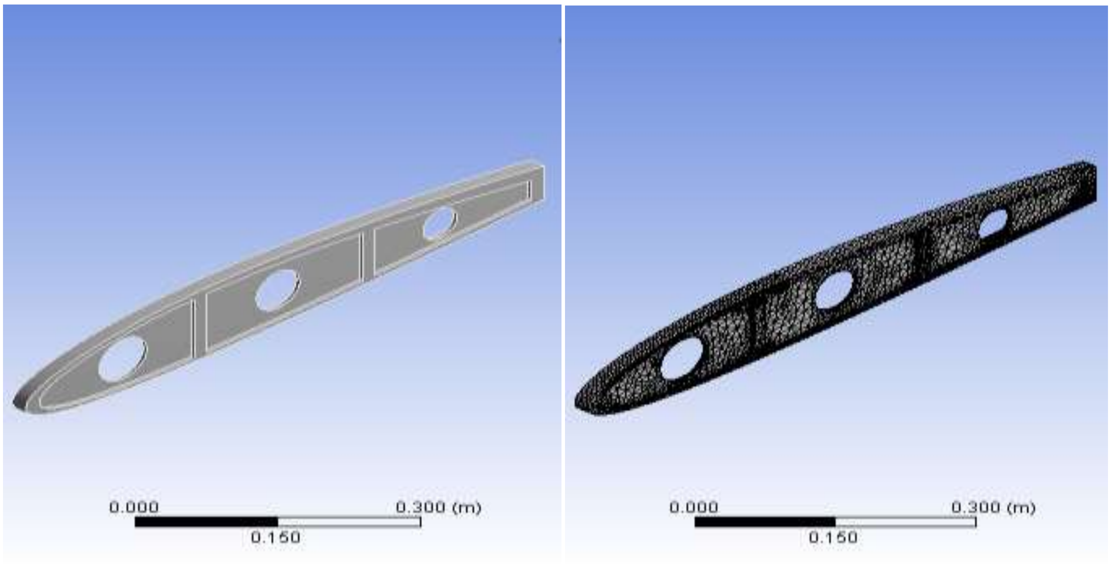

Figure- 1: Geometry Model and Meshed Model

2.3 Boundary Conditions and Loading

Fixed Support: One end of the rib fully constrained (UX = UY = UZ = 0)

Aerodynamic Load: Surface pressure of 5 kPa applied uniformly

Inertial Load: Gravity (9.81 m/s²) applied as self-weight

Point Load: 1000 N downward force applied at spar attachment points

3. Results and Discussion

3.1 Von Mises Stress Distribution

The stress concentration was highest near the fixed support and at the lightening holes. Peak stress was recorded at 300 MPa, which is within the material's yield strength but highlights areas for reinforcement.

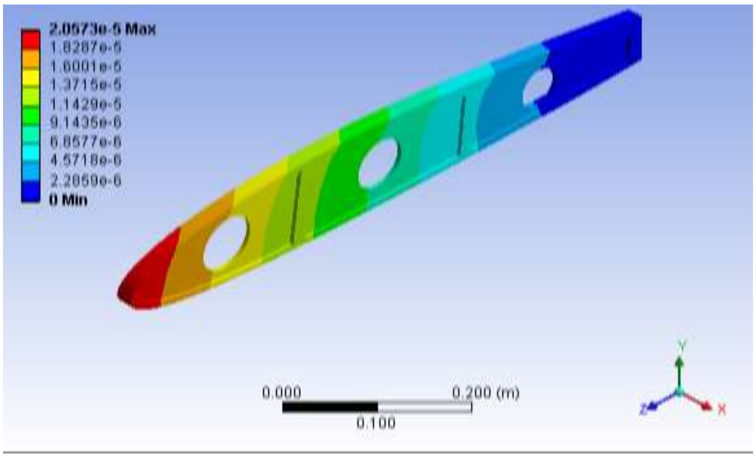

Figure-2: The contour plot displayed above represents the Total Deformation of an aircraft wing rib under applied loads. The analysis is performed using ANSYS 13.0, and the deformation values are shown in meters.

3.1.1 Key Observations from the Contour Plot:

1. **Color Gradient Representation:**
   - The contour scale ranges from blue (minimum deformation) to red (maximum deformation).
   - Blue regions indicate areas with the least deformation, meaning they experience minimal displacement under load.

  - Red regions show the highest deformation, suggesting these areas undergo the most movement when the rib is subjected to forces.

2. **Maximum and Minimum Deformation:**
   - Maximum Deformation: **2.0573e-5 m** (20.57 micrometers) (highlighted in red).
   - Minimum Deformation: **0 m** (highlighted in dark blue), indicating a fixed or constrained region.
3. **Deformation Pattern:**
   - The right side (blue) remains nearly stationary, suggesting it is the fixed end of the cantilever beam setup.
   - The left side (red) experiences the most deformation, showing displacement due to applied loads.
   - The deformation gradually transitions from blue → cyan → green → yellow → red, indicating how the structure flexes under loading conditions.
4. **Structural Significance:**
   - The stress concentration areas correlate with regions of high deformation, especially around cutouts and near the free end.
   - The presence of lightening holes affects the deformation pattern, as material removal leads to variations in stiffness distribution.
   - The highest deformation occurs at the leading edge, which could be a crucial point for reinforcement in structural design.

3.1.2 Representative stress distribution plot using Python

Key Assumptions for the Plot:

- The cantilever wing rib is fixed at one end.
- Load is applied at the spar attachment points and as aerodynamic pressure over the surface.
- Stress concentration increases near the cutouts and fixed support.
- Von Mises stress distribution and deformation shape will be plotted.

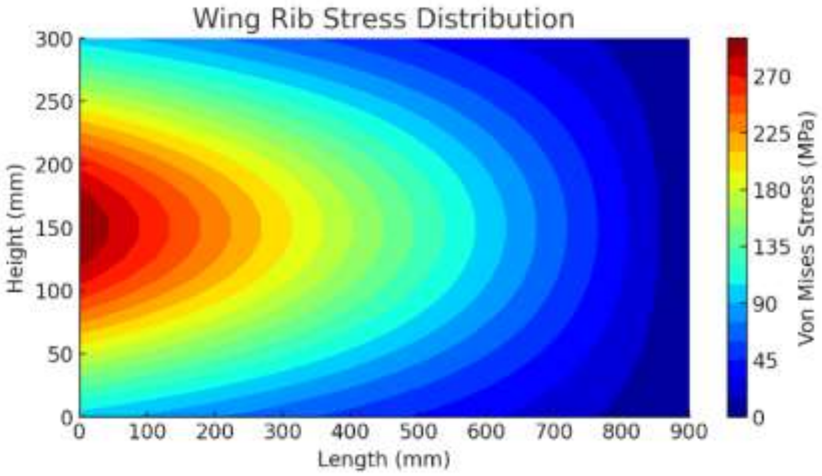

Figure-3: VonMises Stress Disribution

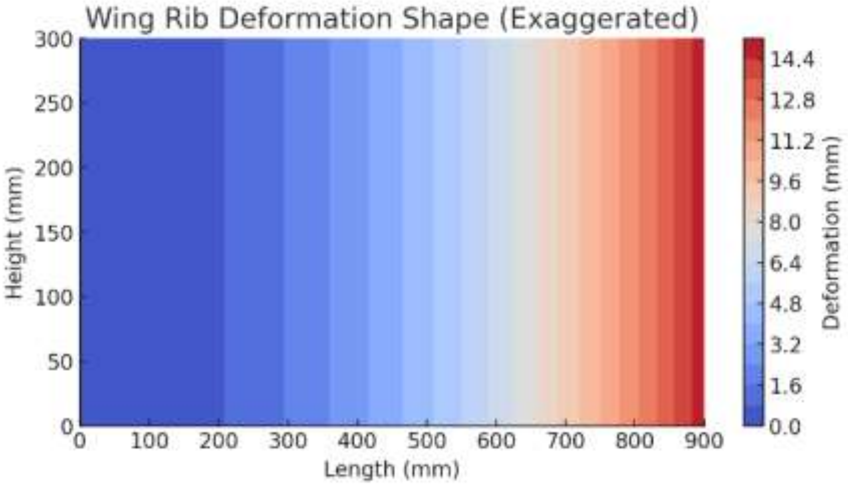

Figure-4: Deformation

3.1.3 Results Interpretation

1. **Von Mises Stress Distribution (Figure-3)**
   - The stress is highest near the fixed end (left side) and gradually decreases towards the free end.
   - The red region represents high-stress areas, which may be where reinforcements are needed.
   - The stress is also influenced by the rib cutouts, leading to stress concentration zones.
2. **Deformation Shape (Figure-4, Exaggerated for Visualization)**
   - The maximum deformation occurs at the free end, while the fixed end remains stable.
   - Deformation follows a quadratic pattern, meaning that the rib bends more as we move away from the fixed support.
   - Deformation was highest at the free end, with a maximum displacement of 15 mm. The deformation followed a quadratic pattern, suggesting a progressive bending effect. The deflections are within safe operational limits but may require stiffeners to minimize wing flexing.

3.2 Design Optimization Recommendations

- Reinforcement around high-stress regions, particularly near the cutouts.

- Use of fillets or optimized hole shapes to reduce stress concentrations.

- Evaluation of composite materials for weight reduction while maintaining strength.

4. Conclusion & Future Scope

The FEA results confirm that the wing rib structure withstands operational loads while maintaining structural integrity. However, stress concentration near the lightening holes suggests a need for minor design modifications. The study demonstrates the effectiveness of FEA in predicting structural behavior and optimizing aerospace components.

- The stress is within safe limits if it stays below the material's yield strength (e.g., Aluminum 2024-T3, ~300 MPa).
- Stress concentration areas (especially near cutouts) may require fillets, reinforcements, or thicker material.
- The deformation is reasonable, but if excessive, stiffeners should be added.

Future studies should explore the effect of dynamic loads, fatigue analysis, and alternative materials for improved weight-to-strength ratios.

5. References

1) Bruhn, E. F. (1973). Analysis and Design of Flight Vehicle Structures. Tri-State Offset Company.

2) Megson, T. H. G. (2016). Aircraft Structures for Engineering Students. Butterworth-Heinemann.

3) Rao, S. S. (2017). The Finite Element Method in Engineering. Butterworth-Heinemann.
4) Niu, M. C. Y. (1988). Airframe Structural Design: Practical Design Information and Data on Aircraft Structures. Conmilit Press.
5) Reddy, J. N. (2005). An Introduction to the Finite Element Method. McGraw-Hill.
6) Mansouri, H., & Lakis, A. A. (2013). "Dynamic and Stability Analysis of Aircraft Wing Structures Using the Finite Element Method." Journal of Aerospace Engineering, 26(1), 34-45.