International Journal of
AI ELECTRICAL CIVIL AND MECHANICAL ENGINEERING
Peer Reviewed, Referred & Indexed Journal
ISSN: 3070-0434 www.zesterapublications.com Original Research Paper

AEROSPACE STRUCTURAL BRACKET WITH STATIC SIMULATION

[1] A Sandhya, [2] E Vaishnavi, [3] Ch Leela, [4] G Arun, [5] B Naresh
[1]AssistantProfessor, [2345]Students
Department of MECH
Siddhartha Institute of Technology & Sciences, Narapally
aitharajusandhya_mech@siddhartha.co.in, 24tq5a0303@siddhartha.co.in,
24tq5a0301@siddhartha.co.in, 24tq5a0331@siddhartha.co.in, 24tq5a0332@siddhartha.co.in,

Abstract

The Aerospace Structural Bracket with Static Simulation project focuses on the design and structural evaluation of a lightweight bracket used in aerospace applications. Aerospace structural components must withstand mechanical loads while maintaining low weight, high strength, and reliable performance. Therefore, proper design and structural analysis are essential to ensure that the bracket can safely operate under the expected loading conditions.

In this project, a detailed 3D CAD model of an aerospace structural bracket is developed by considering important design parameters such as dimensions, mounting holes, material properties, thickness, fillets, and load-bearing regions. The geometry is designed to provide adequate strength while minimizing unnecessary material and weight.

A static structural simulation is performed on the developed bracket to evaluate its behavior under applied loads and boundary conditions. The analysis can provide important results such as Von Mises stress, total deformation, strain, and factor of safety. These results help identify critical regions where maximum stress or deformation occurs and determine whether the proposed design can safely withstand the applied loading conditions.

The simulation results can be used to validate the structural integrity of the bracket and identify areas that require design improvement. Design parameters such as thickness, fillet radius, material, and geometry can be modified and analyzed again to achieve an optimized structure.

I. Introduction

An aerospace structural bracket is an important mechanical component used to support, connect, and secure various structures and equipment in aircraft and aerospace systems. These brackets are subjected to different mechanical loads during operation, including tension, compression, shear, vibration, and external forces. Therefore, the bracket must have sufficient strength and stiffness to safely withstand the applied loads.

Aerospace components are generally designed with a strong focus on weight reduction and structural reliability. Reducing unnecessary material can decrease the overall weight of an aircraft and improve fuel efficiency, while maintaining adequate strength is essential for safe operation. As a result, aerospace brackets require careful

Received: 24-07-2026 | Accepted: 30-08-2026 | Published: 05-09-2026

International Journal of
AI ELECTRICAL CIVIL AND MECHANICAL ENGINEERING
Peer Reviewed, Referred & Indexed Journal
ISSN: 3070-0434 www.zesterapublications.com Original Research Paper

consideration of their geometry, material, thickness, mounting arrangement, and load-bearing regions.

In this project, a detailed 3D CAD model of an aerospace structural bracket is developed using suitable design dimensions and structural requirements. Features such as mounting holes, ribs, fillets, and curved surfaces are incorporated to improve the strength and functionality of the bracket. Proper geometric design also helps reduce stress concentrations at critical locations.

The developed bracket is evaluated using static structural simulation to understand its response under specified loading conditions. Appropriate material properties, constraints, and external loads are applied to the model. The simulation provides important results such as Von Mises stress, total deformation, strain, and factor of safety.

II. Literature Survey

Several researchers have studied aerospace structural brackets with the objective of achieving high strength, low weight, and reliable performance under different loading conditions. Aerospace brackets are commonly designed to support or connect structural components, so their geometry and material selection play an important role in maintaining structural integrity. Traditional design approaches mainly depend on analytical calculations and experimental testing to determine whether a bracket can safely withstand the expected loads.

Previous studies have investigated the use of finite element analysis (FEA) for evaluating aerospace structural components. FEA allows engineers to predict stress, strain, deformation, and other structural responses before manufacturing a physical prototype. Static structural analysis is particularly useful during the preliminary design stage because it helps identify critical regions and potential failure locations.

Research has also focused on lightweight structural design for aerospace applications. Reducing component mass while maintaining sufficient strength can improve overall aircraft efficiency. Techniques such as topology optimization, material reduction, and improved geometric design are commonly used to remove unnecessary material from low-stress regions while retaining material in highly loaded areas.

The selection of aerospace materials is another important research area. Materials such as aluminum alloys, titanium alloys, and high-strength composite materials are considered for different structural applications based on their strength-to-weight ratio, stiffness, corrosion resistance, and operating environment. Proper material selection can significantly influence the structural performance and weight of a bracket.

Researchers have also examined the influence of geometric features such as fillets, ribs, mounting holes, and thickness variations on bracket performance. Sharp corners and sudden changes in geometry can produce stress concentrations, while properly designed fillets and reinforcing ribs can improve load distribution and reduce localized stresses.

Recent studies increasingly combine CAD modeling, FEA, and optimization techniques to develop aerospace components more efficiently. A three-dimensional CAD model can be analyzed under different loading conditions, and the simulation results can be used to modify the geometry before manufacturing. This reduces the requirement for repeated physical prototypes and helps shorten the design-development cycle.

Based on the existing research, the proposed Aerospace Structural Bracket with Static Simulation project focuses on creating a detailed 3D CAD model and evaluating its structural behavior under static loading. The analysis can determine Von Mises stress, total deformation, strain, and factor of safety. The results can be used to identify critical regions and optimize parameters such as thickness, fillet radius, material, and structural features to obtain a lightweight, strong, and reliable aerospace bracket.

III. System Analysis

System analysis is an important stage in the development of the Aerospace Structural Bracket with Static Simulation project. The analysis focuses on understanding the structural requirements and loading conditions of the bracket. Important parameters such as bracket dimensions, material properties, thickness, mounting locations, and applied loads are considered. The geometry is designed to provide sufficient strength while minimizing unnecessary weight. Features such as mounting holes, fillets, ribs, and support regions are incorporated into the design. A detailed three-dimensional CAD model of the bracket is created using suitable modeling software. The developed model is checked for dimensional accuracy and possible geometric errors. Appropriate boundary conditions are identified based on how the bracket is mounted in the actual application. Static loads are applied to the required regions of the model for structural evaluation. Finite element analysis is used to calculate stress, deformation, strain, and factor of safety. Critical regions experiencing high stress or deformation are identified from the simulation results. Overall, the system analysis provides the necessary foundation for developing a lightweight, strong, and reliable aerospace structural bracket.

Existing System

In the existing system, aerospace structural brackets are commonly designed using conventional calculations, CAD modeling, and physical prototype testing. Engineers initially determine the required dimensions and material based on expected loading conditions. The bracket geometry is then developed using traditional design methods and evaluated through analytical calculations. Physical prototypes may be manufactured to verify the strength and functionality of the designed component. Manufacturing and testing multiple prototypes can increase development time and production costs. Conventional calculations may not provide detailed information about the complete stress distribution throughout the bracket. Identifying local stress concentrations around holes, corners, and mounting regions can also be difficult. Design modifications often require repeated calculations and prototype manufacturing. The effect of geometric changes on stress and deformation may therefore take considerable time to evaluate. Traditional testing also requires suitable equipment, testing facilities, and resources. These limitations can make the design optimization process slower and more expensive. Therefore, simulation-based structural analysis provides a more efficient approach for evaluating aerospace bracket designs before manufacturing.

Disadvantages of Existing System

- Requires extensive manual calculations.
- Physical prototypes may be required for validation.
- Prototype manufacturing increases development cost.
- Testing can be time-consuming.
- Difficult to visualize internal stress distribution.
- Local stress concentrations may be difficult to identify.
- Design modifications require repeated calculations.
- Multiple prototypes may be needed for optimization.
- Limited early-stage deformation analysis.
- Requires specialized testing equipment.
- Difficult to evaluate many design alternatives quickly.
- Weight optimization can be challenging.

Proposed System

The proposed system focuses on designing and evaluating an aerospace structural bracket using CAD modeling and static structural simulation. The design begins by identifying the required dimensions, material properties, mounting conditions, and applied loads. A three-dimensional bracket model is created with suitable structural features such as mounting holes, fillets, ribs, and support sections. The geometry is optimized to provide sufficient strength while reducing unnecessary material and overall weight. The completed CAD model is transferred to a finite element analysis environment for structural simulation. Suitable material properties are assigned to the bracket according to the selected aerospace material. Boundary conditions are applied at the mounting regions to represent the actual support conditions. Static loads are applied to the appropriate areas of the bracket. The simulation calculates important results such as Von Mises stress, total deformation, strain, and factor of safety. Critical regions are identified from the obtained simulation results. If excessive stress or deformation is observed, the bracket geometry can be modified and analyzed again. Overall, the proposed system provides an efficient approach for developing a strong, lightweight, and reliable aerospace structural bracket.

Advantages of Proposed System

- Provides accurate 3D CAD modeling.
- Reduces dependence on physical prototypes.
- Saves design and development time.
- Reduces testing and manufacturing costs.
- Provides detailed stress distribution.
- Identifies critical stress concentration regions.
- Shows total deformation and strain.
- Calculates factor of safety.
- Supports lightweight design.

- Allows easy modification of geometry.
- Supports design optimization.
- Improves structural reliability.

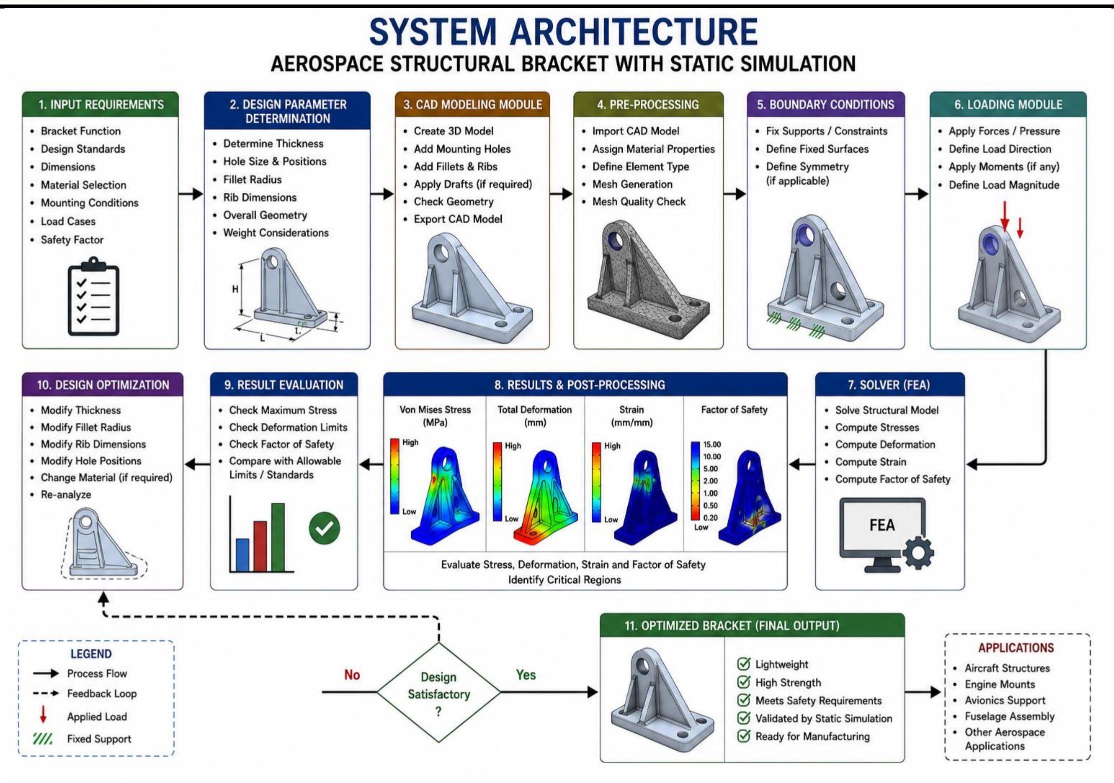
IV. Methodology

The methodology begins by identifying the structural requirements of the aerospace bracket, including dimensions, loading conditions, mounting arrangements, and material properties. Suitable aerospace material is selected according to the required strength, stiffness, and weight considerations. A three-dimensional CAD model of the bracket is developed with appropriate thickness, mounting holes, fillets, ribs, and support features. The completed geometry is checked for accuracy and prepared for finite element analysis. The CAD model is imported into suitable simulation software and the material properties are assigned. The geometry is divided into a suitable finite element mesh to perform numerical calculations. Appropriate boundary conditions are applied to the mounting regions of the bracket. Static loads are then applied according to the expected operating conditions. The solver performs the structural analysis and calculates stress, deformation, strain, and factor of safety. The simulation results are examined to identify critical regions and potential structural problems. Based on the results, design parameters such as thickness, fillet radius, ribs, or material can be modified. Finally, the improved design is analyzed again to verify its structural performance and suitability for the intended aerospace application.

System Architecture

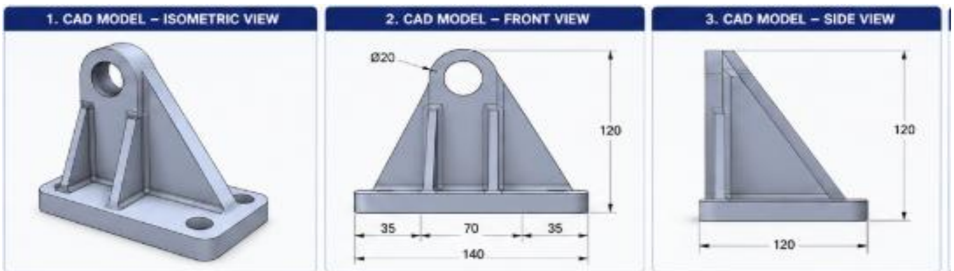
The system architecture represents the complete workflow used for designing and analyzing the aerospace structural bracket. The process begins with input requirements such as bracket dimensions, material properties, mounting conditions, and expected loads. These inputs are provided to the design parameter module for determining suitable geometric specifications.

Zestera PUBLICATIONS
International Journal of
AI ELECTRICAL CIVIL AND MECHANICAL ENGINEERING
Peer Reviewed, Referred & Indexed Journal
ISSN: 3070-0434 www.zesterapublications.com Original Research Paper

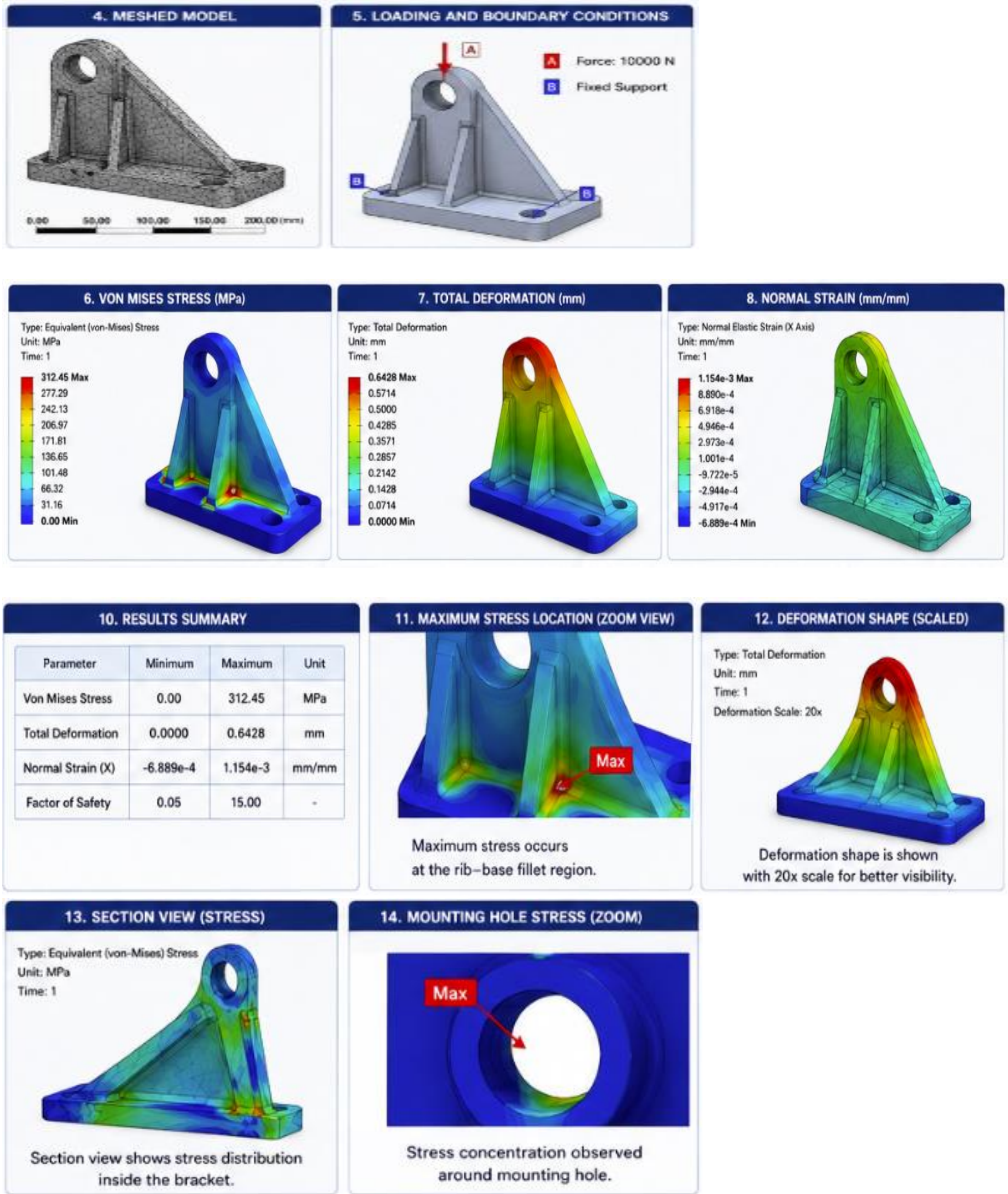


The CAD modeling module creates the three-dimensional bracket geometry with required holes, fillets, ribs, and support features. The geometry is then transferred to the preprocessing stage for material assignment and mesh generation. The boundary condition module applies suitable constraints to represent the actual mounting conditions of the bracket. The loading module applies the required static forces to the appropriate regions of the model. The finite element solver performs numerical calculations to determine the structural response of the bracket. The post-processing module generates Von Mises stress, deformation, strain, and factor-of-safety results. The results are analyzed to identify critical regions and evaluate the structural safety of the design. If the results do not satisfy the required conditions, the design parameters are modified and the simulation is repeated. The optimized model is then finalized after meeting the required strength and weight criteria.

V. Result and Output



International Journal of
AI ELECTRICAL CIVIL AND MECHANICAL ENGINEERING
Peer Reviewed, Referred & Indexed Journal
ISSN: 3070-0434 www.zesterapublications.com Original Research Paper



Parameter	Minimum	Maximum	Unit
Von Mises Stress	0.00	312.45	MPa
Total Deformation	0.0000	0.6428	mm
Normal Strain (X)	-6.889e-4	1.154e-3	mm/mm
Factor of Safety	0.05	15.00	-

ANALYSIS DETAILS

Analysis Type	:	Static Structural
Solver	:	Mechanical APDL
Element Type	:	Solid (Tetrahedral)
Material	:	Aluminum Alloy 7075-T6
Nodes	:	145,678
Elements	:	82,341

VI. Conclusion

The Aerospace Structural Bracket with Static Simulation project successfully demonstrates the design and structural evaluation of an aerospace bracket using CAD modeling and finite element analysis. The bracket is designed with suitable geometric features such as mounting holes, fillets, ribs, and support sections to provide adequate strength while minimizing unnecessary weight.

Static simulation helps evaluate the behavior of the bracket under the applied loading conditions. The analysis provides important results such as Von Mises stress, total deformation, strain, and factor of safety, which help identify critical regions and assess the structural safety of the design.

The simulation results can be used to identify areas of high stress and deformation and make necessary improvements to the bracket geometry. Design parameters such as thickness, fillet radius, rib dimensions, and material can be modified to achieve a better strength-to-weight ratio.

Overall, the project demonstrates that combining CAD modeling with static structural simulation is an effective approach for designing a lightweight, strong, and reliable aerospace structural bracket. The developed model can be further improved through topology optimization, fatigue analysis, dynamic analysis, and experimental testing before practical manufacturing and aerospace application.

References

[1] Kumar, R. D., Prudhviraj, G., Vijay, K., Kumar, P. S., & Plugmann, P. (2024). Exploring COVID-19 through intensive investigation with supervised machine learning algorithm. In Handbook of Artificial Intelligence and Wearables (pp. 145-158). CRC Press.
[2] Swathi, B., Vijay, K., Sushanth Babu, M., & Dinesh Kumar, R. (2024, November). Machine Learning Techniques in Cloud Based Intrusion Detection. In The International Conference on Artificial Intelligence and Smart Environment (pp. 557-564). Cham: Springer Nature Switzerland.
[3] Sv satyakrishna, shirisha rangu ,bhargavi nalacheruve.(2024) Prospective investigation on colorectal cancer with SMOTE on machine learning Algorithm
[4] Dr.G.Vishnu Murthy, BhargaviNalacheruve 1Professor, Department of computer Science & engineering, Anurag University, TS, India. 2Student, Department of computer Science & engineering, Anurag University, TS, India.
[5] V. N. S. Manaswini, K. K, C. Nigam, S. S. Ali, R. Niranjana, and Suman, "Real-Time Object Detection in Drone Surveillance Using YOLOv5," in Proc. 2025 3rd Int. Conf. IoT, Communication and Automation Technology (ICICAT), Gorakhpur, India, 2025, pp. 1–6, doi: 10.1109/ICICAT68430.2025.11414670.
[6] B. Soundarya, V. N. S. Manaswini, M. Ayyakrishnan, R. D. Kumar, "Contextual Analysis of Big Data Analytics in Intelligent Transportation Frameworks," in Intersection of Artificial Intelligence, Data Science, and Cutting-Edge Technologies: From Concepts to Applications in Smart Environment, Lecture Notes in Networks and Systems, vol. 1353, Cham: Springer, 2025, doi: 10.1007/978-3-031-88304-0_79.
[7] R. D. Kumar, V. N. S.Manaswini, "Applications of blockchain in smart cities: detecting fake documents from land records using blockchain technology," in Blockchain for Smart Cities, Elsevier, 2021, pp. 105–117, doi: 10.1016/B978-0-12-824446-3.00017-X.
[8] Tejavath Veeramma, Badarla Anil, Guguloth Ravinder, "An advanced movie recommender using collaborative filtering and sentiment analysis," International

Research Journal of Modernization in Engineering Technology and Science, vol. 7, no. 7, July 2025, doi: 10.56726/IRJMETS81618.

[9] Ravi Kumar Banoth, Ramana Murthy B V, "Automatic crop recommendation system using LightGBM and decision tree machine learning models," Journal of Machine and Computing, vol. 5, no. 1, pp. 343, Jan. 2025, doi: 10.53759/7669/jmc202505026.

[10] Ravi Kumar Banoth, Dr. B.V. Ramana Murthy, "Smart agriculture through IoT and machine learning for analyzing carbon footprints," in Proc. Int. Conf. Computer Science and Communication Engineering (ICCSCE), Apr. 2025.

[11] Ravi Kumar Banoth, B. V. Ramana Murthy, "Soil image classification using transfer learning approach: MobileNetV2 with CNN," SN Computer Science, vol. 5, art. no. 199, 2024, doi: 10.1007/s42979-023-02500-x.