ISSN: 1813-4890

Mechanical Analysis and Optimization of a Deployable Mechanism for a Space-Extending Arm

Wenjun Zhang, Yushuang Feng *, Bo Yuan and Jiajun Su

School of Mechanical Engineering, Tianjin University of Technology and Education, Tianjin 300222, China

Abstract

With the advancement of aerospace technology, extravehicular activities (EVAs) have become a crucial part of space missions. Astronauts conducting EVAs place increasingly strict requirements on the reliability and flexibility of robotic arm systems. To meet the in-orbit extension requirements of space robotic arms, a deployable mechanism for extending arm segments is proposed, along with its structural modeling and performance evaluation. A 3D model of the deployable mechanism and extension arm was developed, and a static analysis was conducted by simulating the forces exerted by astronauts during EVAs. Simulation results show that under normal loading conditions, the structural strength of the mechanism meets material requirements and exhibits redundancy, indicating good reliability. Based on this analysis, topology optimization was applied to reduce the weight of the deployable mechanism. After optimization, the total weight decreased by 1.8 kg, and the yield stress was reduced from 189.29 MPa to 173.84 MPa. This study offers theoretical support and engineering guidance for the reliable and lightweight design of deployable mechanisms in space robotic arms.

Keywords

Deployable Mechanism, Topology Optimization, Finite Element Simulation, Lightweight Design.

1. Introduction

The space station robotic arm is essential for the assembly, construction, and operational maintenance of space stations. It performs a wide range of tasks, including module relocation and docking, assisting astronauts in installing extravehicular equipment, and maintaining and repairing external components. As space technologies advance and space activities become more frequent, the number of station modules is increasing, along with the demand for equipment maintenance and replacement. To address this need, a deployable mechanism for folding and unfolding robotic arm segments is proposed to assist astronauts with the alignment, fixation, and disassembly of arm components during EVAs[1-2].Currently, a wide range of deployable mechanisms have been developed worldwide. Based on their actuation source, these mechanisms are categorized as actively driven, which rely on electric power, and passively driven, which are entirely mechanical. Early examples of satellites using active deployable mechanisms include NASA’s ECHO I and EXPLORER IX. After the successful Inflatable Antenna Experiment (IAE) in 1996, L’Garde Inc. and NASA’s Jet Propulsion Laboratory (JPL) advanced the use of inflatable deployable mechanisms in antenna drive systems and synthetic aperture radar platforms[3-4]. Passive folding mechanisms typically use spring-driven actuation and are representative of spring-based deployable structures. These systems consist mainly of hinge-connected components, with mechanical power transmitted to the hinges to enable extension or contraction, thereby deploying or retracting structures such as antennas[5].

This study presents a structural design developed with full consideration of the maximum manual force astronauts can exert during on-orbit operations and its impact on the performance of the deployable mechanism. Detailed mechanical simulations were conducted to accurately assess the force distribution and structural deformation of the mechanism under representative loading conditions. Structural optimization and material reduction were applied to critical load-bearing components to ensure sufficient load capacity, mechanical strength, and long-term reliability. The optimized design not only improves the operational efficiency of the mechanism in the harsh space environment but also significantly enhances its safety and adaptability to mission requirements.

2. Structural Design and Mechanical Analysis of the Deployable Mechanism

2.1. Structural Design of the Deployable Mechanism

The deployable mechanism comprises a differential assembly and a deployment actuator module. Power is transmitted through the input shaft of the differential, which drives the actuator module to rotate, enabling the extension arm to fold and unfold. Deployable hinges are connected to the extension arm segments using captive screws. Torque is transmitted between the two hinges via a shaft and a splined sleeve. The differential delivers the driving force, rotating the shaft and allowing the two arm segments to transition from a folded configuration to a fully deployed state, thereby completing the arm extension.

This deployable device is designed for use in the extravehicular environment of a space station to extend robotic arm segments. During operation, the structure is primarily subjected to axial thrust exerted by the astronaut and the rotational inertia generated during deployment. However, due to spatial constraints, the rotation is slow, and the resulting inertial force is minimal. Although this inertia influences the internal transmission design of the differential, its effect on the mechanical structure of the deployable section is negligible. Therefore, only the astronaut-applied axial thrust is considered in the simulation analysis. Based on the results, structural optimization is performed to reduce overall weight and achieve a lightweight design.

2.2. Preprocessing for Finite Element Analysis

In the finite element analysis, tetrahedral solid elements were used to mesh both the deployable mechanism and the extension arm. To ensure both accuracy and computational efficiency, particular attention was paid to the following aspects during meshing:

(1) Avoidance of slender triangular elements and abrupt changes in element size; (2) Application of local mesh refinement in fillets and stress concentration regions. The final meshing result is shown in Figure 1.

During the meshing process in ANSYS, both the deployable mechanism and the extension arm were meshed simultaneously, with mesh quality controlled according to the aforementioned criteria. However, in the subsequent force response analysis and structural optimization, only the deployable mechanism was included.

In terms of material allocation, to ensure consistency between the finite element simulation and the actual engineering application, steel is used for the extension arm, shaft sleeve, and drive shaft, while the remaining components are made of aluminum alloy. The material properties are shown in Table 1.

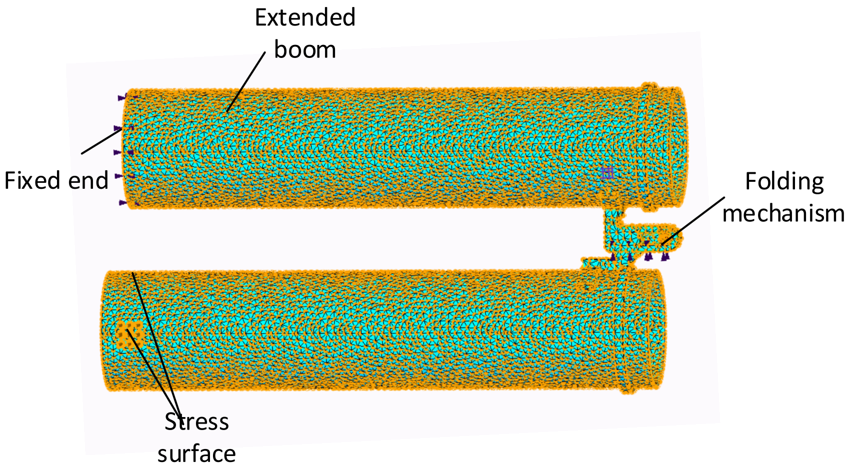


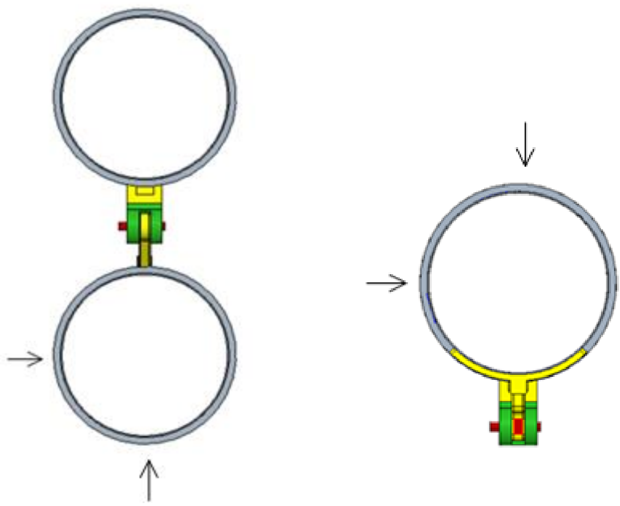
Figure 1: Mesh Partitioning Diagram

Table 1: Material Properties

material	density/(kg/m3)	Young's Modulus/Gpa	Yield strength/Mpa	Poisson's ratio
Aluminum alloy	2700	75	240	0.33
steel	7827	210	235	0.27

2.3. Mechanical Analysis

Considering actual working conditions, the extension process is performed by a single astronaut. Therefore, one arm segment is fixed first, and the connection between the two segments is completed by rotating the differential rocker. In the constraint and load analysis, one side of the arm segment is fixed, while the load is applied to the edge of the other segment. This setup simulates the maximum thrust exerted by the astronaut during installation, which affects the overall structure. The maximum output force exerted by the astronaut in orbit is 88 N, which is used as the applied load on the arm segment. The force application conditions for both scenarios are illustrated in Figure 2.

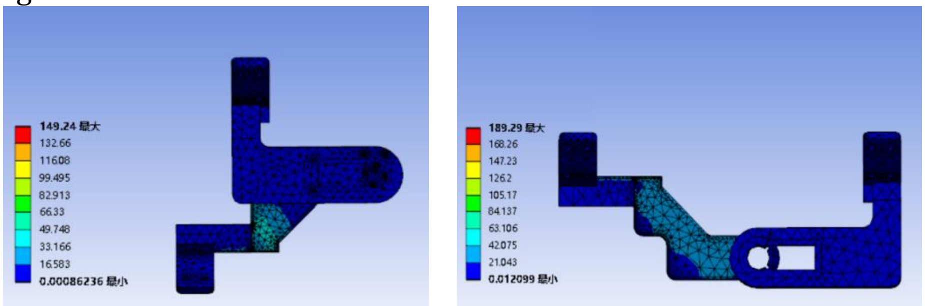


(a)Stressed in folded state (b) Stressed in unfolded state

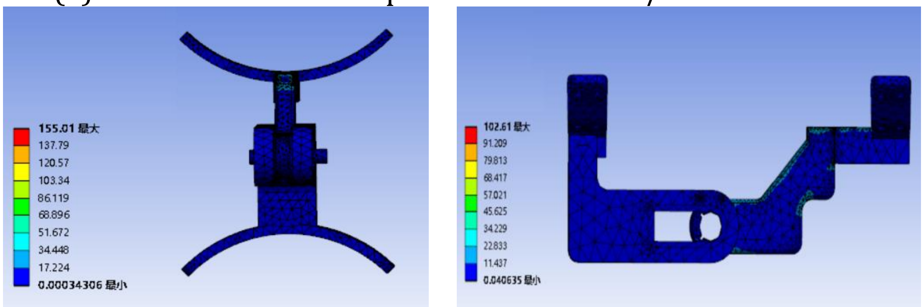
Figure 2: Stress states under two working conditions

ISSN: 1813-4890

After applying the setup as described above, the computational analysis was performed. The stress responses of the initial model under both folding and unfolding conditions are obtained, as shown in Figure 3.



(a) Horizontal stress response in the folded/unfolded state



(b) Vertical stress response in folded/unfolded state

Figure 3: Stress Response under Two Working Conditions

As shown in the figure above, after fixing one arm segment and applying a vertical inward force of 88 N to the other arm segment, the maximum stress in the deployable working section is 149.24 MPa and 189.29 MPa, respectively, both of which are below the ultimate yield strength of the aluminum alloy. The stress concentration primarily occurs at the corner of the deployable hinge, while the stress response on the other side is relatively small, with negligible impact.

3. Optimized Design

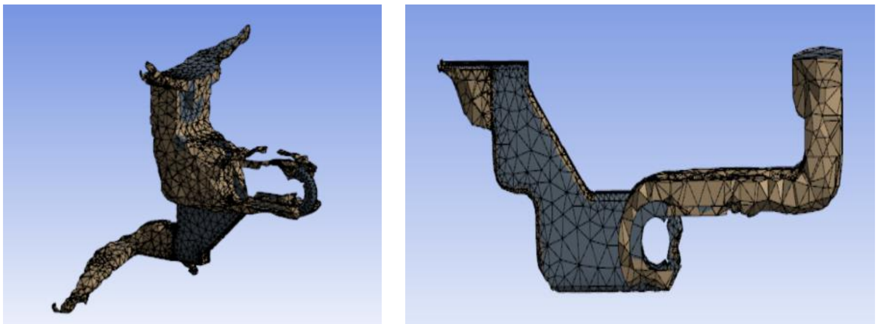
Through the analysis of the simulations under both working conditions, it is evident that the original model offers potential for lightweight design optimization. Excess material can be topologically optimized to reduce weight, while areas with high stress can be further optimized. This allows for a reduction in weight during the transportation process without compromising the normal operation of the mechanism.

Based on the finite element analysis results of the original model, topological optimization is applied to analyze the material distribution of the bracket. Subsequently, a localized optimization and lightweight design scheme is developed using 3D modeling software, considering the actual working conditions, followed by finite element validation.

3.1. Topological Optimization and Analysis

Based on the working conditions and the simulation analysis results mentioned above, while also considering the actual installation and assembly relationships, the model is divided into design and non-design domains. The extension arm and connection parts are not subjected to optimization, and only the deployable mechanism is topologically optimized. The optimization is performed with a 30% weight reduction constraint as the response. The optimization results are shown in Figure 4.

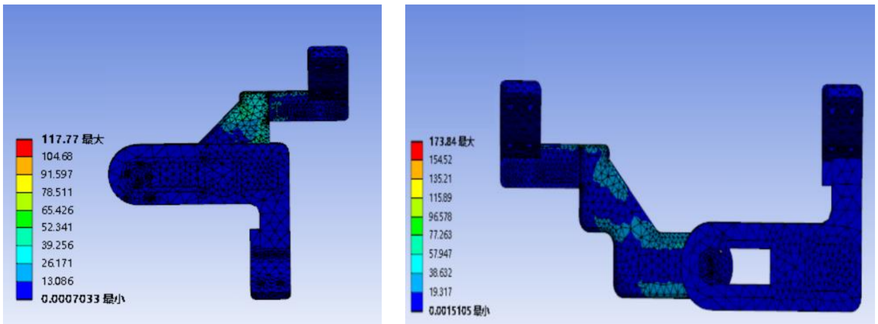
ISSN: 1813-4890



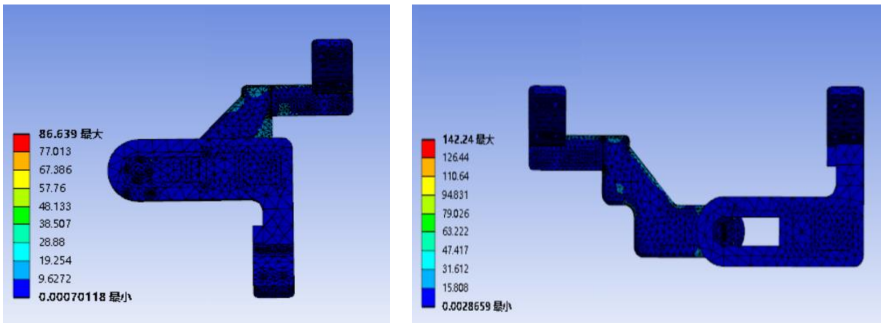
(a) Topology Optimization in Folded State (b) Topology Optimization in Deployed State

Figure 4: Topology Optimization Results

Based on the above optimization results and taking into account the actual working conditions, the original model was redesigned accordingly. To ensure the reliability of the lightweight scheme, finite element verification under both working conditions was conducted for the redesigned structure. The simulation results are shown in Figure 5.



(a) Horizontal stress response in folded/unfolded state



(b) Vertical stress response in folded/unfolded state

Figure 5: Stress response after weight reduction optimization

The simulation results indicate that, after localized and overall lightweight optimization, the maximum stress experienced by the deployable structure is 173.84 MPa, which is lower than the original design's 189.29 MPa and remains below the yield strength. Meanwhile, the total weight is reduced by 1.8 kg. This demonstrates that the weight reduction objective was achieved without compromising the structural performance under normal operating conditions.

3.2. Analysis of Optimization Results

When the deployable mechanism is in a fully extended state and subjected to an external horizontal load, the equivalent stress within the structure reaches its peak. To evaluate the mechanical response of the structure under this extreme loading condition, a static analysis was conducted using the finite element method. The simulation results show that, prior to structural optimization, the maximum equivalent stress in the deployable mechanism was 189.29 MPa.

To improve the structural safety margin and achieve weight reduction, a topology optimization was performed, focusing on stress distribution balance and material removal. During the optimization process, the critical load-bearing paths were preserved, while non-load-bearing regions were reconstructed. As a result, the maximum equivalent stress was reduced to 173.84 MPa, representing an approximate 8.7% decrease compared to the original design, as shown in Table 2. In addition, the total structural mass was reduced from 15.7 kg to 13.9 kg, achieving a weight reduction of 11.46%.

Table 2: Data Comparison

	Horizontal Load in Folded State	Vertical Load in Folded State	Vertical Load in Deployed State	Horizontal Load in Deployed State
Before Optimization	149.24	155.01	102.61	189.29
After Optimization	117.77	86.639	142.24	172.84

Although topology optimization primarily aims to reduce weight, it may also weaken the structure's load-bearing capacity in the event of local damage or functional degradation. Therefore, it is necessary to evaluate the fault-tolerance characteristics of the optimized structure. The analysis results indicate that the optimized structure retains multiple load transmission paths along critical stress-bearing routes, providing a certain degree of structural redundancy.

In addition, significant changes in structural form occurred during the optimization process. Prior to optimization, the deployable mechanism adopted a conventional solid design with relatively uniform material distribution, but with evident redundant regions, leading to excessive overall weight and unclear force transmission paths. After applying topology optimization under defined stress and deformation constraints, the structure gradually evolved into a truss-like layout with clearer load paths. Redundant material was effectively removed, and only essential solid structures were retained at key stress-concentrated locations. This resulted in reduced structural mass without compromising mechanical performance.

The above results clearly demonstrate that the proposed optimization method significantly mitigates stress concentration and enhances the mechanical performance of the structure under extreme loading conditions. Furthermore, the method achieves over 11% mass reduction while ensuring structural safety and stiffness, highlighting its excellent lightweight potential. This enhancement greatly improves the adaptability and reliability of the structure in space environments, showcasing strong engineering applicability and broad prospects for practical implementation.

4. Conclusion

This paper primarily focuses on the structural design, strength analysis, and optimization of a deployable mechanism for space applications, with an in-depth investigation of its mechanical response under typical loading conditions. The stress state and distribution of the deployable mechanism under controlled loading were simulated, followed by structural optimization. Under the premise of satisfying functional and strength requirements, weight reduction was achieved through topology optimization, demonstrating a robust lightweight design approach. The main findings of this study are as follows:

A novel space-assisted deployable mechanism was designed, featuring excellent deployability and spatial coordination characteristics. It assists astronauts in performing folding, deployment, and alignment operations for the extension arm during space missions, significantly improving the efficiency and reliability of in-orbit assembly operations.

The effectiveness and feasibility of the designed structure were validated through finite element analysis (FEA). Based on these results, topology optimization was applied to reduce the structure's weight. After optimization, the total mass was reduced by approximately 1.8 kg, and the maximum stress decreased from 189.29 MPa to 173.84 MPa, effectively mitigating stress concentration. This optimization not only improved structural lightweighting but also enhanced mechanical performance and engineering applicability.

References

[1] Wang D, Liu Z, Tang C. Topological optimization design of multi phase materials under multiple operating conditions[J]. Journal of Physics:Conference Series,2025,2951(1): 012142.

[2] Ma X, Li T, Ma J, et al: Recent advances in space-deployable structures in China, Engineering, Vol. 17(2022)No.10, p.207-219.

[3] Anonymous: Review of space deployable antenna mechanisms, Journal of Mechanical Engineering, Vol. 56(2021)No.5, p.1. DOI:10.3901/JME.2022.05.001.

[4] Guerci J, Jaska E: ISAT-innovative space-based-radar antenna technology, IEEE International Symposium on Phased Array Systems and Technology (2019), p.45-51.

[5] Ma Y, Li Z, Wei Y, et al: Topology optimization with explicit components considering stress constraints, Applied Sciences, Vol. 14(2024)No.16, p.7171.

[6] Wang X, Xie Z, Zhou X, et al: Adaptive path planning for the gantry welding robot system, Journal of Manufacturing Processes, Vol. 81(2022), p.386-395.

[7] Wang X, Gao J, Zhou X, et al: Path planning for the gantry welding robot system based on improved RRT, Robotics and Computer-Integrated Manufacturing, Vol. 85(2024), p.102643.

[8] Wang R, Sun J, Dai J: Design analysis and type synthesis of a petal-inspired space deployable-foldable mechanism, Mechanism and Machine Theory, Vol. 141(2023), p.151-170.

[9] Guerci J, Jaska E: ISAT-innovative space-based-radar antenna technology, IEEE International Symposium on Phased Array Systems and Technology (2022), p.45-51.

[10] Zhao L, Yang Q, Zhang F, et al: Topology optimization design of rocket bracket structure based on ANSYS Workbench, Computer Aided Engineering, Vol. 33(2024)No.2, p.6-9.

[11] Li H, Hao J, Ding H, et al: Strength and modal analysis of vibrating screen frame using ANSYS, Grain Processing, Vol. 49(2024)No.5, p.50-54.

[12] Ma X F, Li Y, Xiao Y, et al: Development and tendency of large space deployable antenna reflector, Space Electronic Technology, Vol. 15(2018)No.2, p.16-26.

[13] Zhou T, Duan W, Hu S B, et al: Multi-view parametric geometry reconstruction method for optimum topologic results, Digital Manufacturing Science, Vol. 17(2019)No.1, p.68-72.

[14] Ida E, Carricato M: Static workspace computation for under-actuated cable-driven parallel robots, Mechanism and Machine Theory, Vol. 193(2024), p.1-19.

[15] Zhao Y T, Liang C Y, Gu Z Z, et al: A new design scheme for intelligent upper limb rehabilitation training robot, International Journal of Environmental Research and Public Health, Vol. 17(2020)No.8, p.2948.