International Core Journal of Engineering Volume 9 Issue 8, 2023
ISSN: 2414-1895 DOI: 10.6919/ICJE.202308_9(8).0018

Design and Finite Element Analysis of Target Star Structure

Ruixin Gao[1], Lei Dong[1, *], Haojie Zhu[1], Jiabin Wang[2], Yongqi Li[1]

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

[2] Tianjin Navigation Instrument Research Institute, Tianjin, 300131, China

*donglеihit@126.com

Abstract

According to the characteristics of the target star, the main body structure of the star is designed as a framework structure. The solar panels are located on both sides, which can be fixed at any angle and have functions such as folding and retracting. According to the requirements, a finite element model of the target star was established, and structural analysis was performed using finite element software, including static analysis, rotational acceleration analysis, collision force analysis, and modal analysis. Under high-speed rotational conditions, the maximum stress occurs at the hinge connection, with a value of 59.348 MPa and a safety margin of 2.5. The rationality of the structural design was evaluated through finite element analysis. The objective of this study is to introduce the structure of the target star and the application of finite element analysis methods, providing design ideas and validation methods for subsequent researchers.

Keywords

Target Star; Structural Design; Finite Element Analysis; Stress.

1. Introduction

The target star is used to simulate the target characteristics of spacecraft in orbit, including the rotational characteristics, external structural characteristics, reflection characteristics towards sunlight, and spatial operational characteristics. This provides a more realistic target for measurements, identification, reconstruction, and operations in orbit. The main purpose of designing the target star is to provide an experimental object for ground experiments.

With the rapid development of the aerospace industry, the structural design and stability of satellites have also become hot topics in research and application. Gu Dequ [1] et al. studied the design methodology and model analysis of satellite communication antenna structures. Li Hang [2] et al. proposed an innovative optimization method that combines topology optimization, dimensional optimization, and biomimetic lattice structures. They applied this method to optimize the support structure of the FY-3 satellite payload. Xu Xin [3] et al. conducted structural stiffness optimization and experimental analysis on the satellite using Patran/Nastran software to meet the stiffness requirements of the rocket. They analyzed and verified the optimized structure, achieving high stiffness and strength characteristics for the satellite. Tao Xiao [4] et al. conducted simulation verification, structural design, and testing of the antenna for the AP-6D satellite. They successfully validated the importance of their design methodology through these processes. Singh Vimlesh [5] et al. designed fractal antennas for the metrology satellite of DSG, achieving multi-frequency characteristics. Mindaro H S [6] et al. designed the structure of Surya Satelit-1 and conducted static analysis, modal analysis, random vibration analysis, and thermal analysis, among others. Yang Jiandong [7] et al. conducted research on small satellites and separation technology, specifically addressing the satellite and arrow separation problem. They discussed the use of elastic metal unlocking devices, which laid a foundation for advancing the technological level in this area. Rianto Puji [8] et al. conducted the structural design of a satellite and, after analysis, determined its structure. He Junwei [9] et al. summarized the research on deployment mechanisms based on biomimetic principles. The results indicate that the expansion mechanism demonstrated by natural organisms has natural advantages in folding optimization and smooth unfolding processes. This research has significant implications for the structural design of space folding mechanisms such as satellite solar panels. Wang Yan [10] et al. proposed a design process for deployable support structures based on modal analysis. They also studied the DST finite element model of a satellite SAR (Synthetic Aperture Radar). Based on this, they analyzed and optimized the influencing factors of the basic frequency and mass, such as geometric parameters, locking positions, hinge stiffness, and truss joint thickness. The research results are of great significance for improving the performance of DST.

Based on the current research status and practical engineering considerations, this paper proposes a structural design of the target star. The main load-bearing structure is a framework structure, with an external cover made of aluminum skin, effectively improving the stiffness and strength of the target star. The solar panels have the capability to be fixed and folded at different angles. Through finite element analysis, the results indicate that the structural design meets the requirements of the target star.

2. The Structural Design of the Target Star

The satellite structure provides a stable platform for the satellite, ensuring the safety and reliable operation of other payload equipment. The satellite structure provides a stable platform for the satellite, ensuring the safe and reliable operation of other payload equipment. The main load-bearing structure of the satellite forms the force transmission path and acts as the "skeleton" of the satellite. It provides support for the satellite body and individual equipment, ensuring the stability of the structural subsystems and improving the accuracy of installation. Currently, the main part of satellite structures both domestically and internationally is the load-bearing structure, which can be classified into four configurations: frame-type, box-beam-type, central load-bearing cylinder-type, and sandwich-type. The frame-type structure adopts a configuration of rods and beams, assembled through bolt and welding connections. It has a well-defined path for force transmission and exhibits good flexibility, providing overall stiffness for the satellite. The box-beam structure divides and supports the internal components of the satellite, forming a closed box-like structure externally. This design helps to increase the overall stiffness of the satellite. The central load-bearing cylinder structure consists of a cylinder and several circular frames, including sandwich and thin-walled reinforced structures. The stacked structure connects multiple boxes through rod-shaped elements, forming a modular system.

This paper mainly focuses on utilizing a frame-type structure as the main load-bearing structure for the target star. The frame-type structure offers clear force transmission paths and exhibits good flexibility, which can effectively enhance the overall stiffness of the star. According to the design requirements, the main body structure of the target star should have sufficient stiffness to withstand the dynamic loads generated by high-speed rolling. It should also meet the strength and total weight requirements of the star body. The main body of the target star is equipped with two solar panels, with each panel consisting of four identical substrates and one rocker arm structure that connects the star body with the substrate. The solar panels have the capability to rotate and fix around a specified axis (X-axis) as well as to fold and unfold. They can rotate at any desired angle and are equipped with a locking mechanism.

The structure of the target star can be divided into two parts: the star body and the solar panel structure. The star body is shown in Figure 1. The star body interior is constructed using a framework model consisting of beams and stiffeners welded together. The exterior is covered with aluminum skin that is riveted onto the framework structure. The internal framework structure is constructed using welded

5A06 aluminum alloy beams with dimensions of 60mm×60mm×3050mm, and a thickness of 10mm. The side panels are equipped with 20 evenly distributed M4 threaded holes along each boundary, allowing them to be connected to the supporting structure. The internal supporting structure is designed with a mechanical arm connection device at the bottom. The bottom plate has dimensions of 1190mm×1060mm and a thickness of 60mm. It is equipped with a flange connection with an outer diameter of 660mm, inner diameter of 600mm, and thickness of 50mm. This allows it to be compatible with the gribbing action of a mechanical arm.

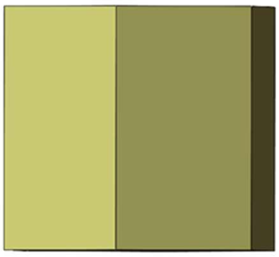

Figure 1. Star body

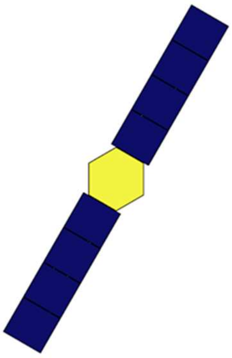

Figure 2. Entire star

The storage container is made from aluminum alloy plates, with multiple layers of insulation attached to the outside, and an optical layer is applied on the outermost surface to ensure its surface reflects light in the same way as a real spacecraft. Each side of the target star's solar sail is composed of three substrates, with each solar sail panel being made up of three substrates. The dimensions of each substrate are 2300mm×2400mm×20mm. In order to increase strength and reduce deformation, an aluminum alloy casing is installed on the outer side of the solar energy solar panel. The casing is perforated to allow for the connection of adjacent substrates using a bolt lock mechanism. This ensures that each substrate can be individually installed and disassembled.

The bolt connection mechanism can be rotated within a range of 0° to 90° using a locking mechanism. It can also be locked in place after full deployment to ensure the stability of the solar panel during operation. The three substrates are connected to the main body of the target star through a connecting frame, which consists of a substrate bolt mechanism and a slider mechanism. The slider mechanism allows for the solar sail panel to move within a range of 0° to 45°. It can also facilitate the deployment and retraction of the solar frame and can be locked in place in any position.

Based on the structural layout of both the target star, the main load-bearing framework and the solar sail panel, the overall material chosen is the aluminum alloy 5A06. Using structural design software, the total mass of the entire star is calculated to be 886.72kg, which satisfies the design requirement of 1000kg.

3. Mechanical Analysis

In order to validate the feasibility of the structural design of the target star, computational analysis is performed using Ansys finite element software to examine the completion of corresponding indicators. To establish a finite element model of the entire target star, it is necessary to simplify the model. The bolt connections between the bottom plate of the main body structure, the side panels, and the main load-bearing structure are assumed to be rigid connections meaning that the finite element nodes of each component are continuous. The hinge connections between the solar sail panel and the main body, as well as between the solar sail panels themselves, are defined using the hinge connection feature in the software to ensure their freedom of movement.

Perform static analysis on the target star ontology of the design, with the main objective being to verify whether the maximum deformation at both ends under the influence of Earth's gravity exceeds 10mm when the solar sail is unfolded. By applying constraints to its bottom and overall gravity, the calculation result is shown in Figure 3. The maximum deformation at both ends is calculated to be 3.59mm. It fufills the requirement.

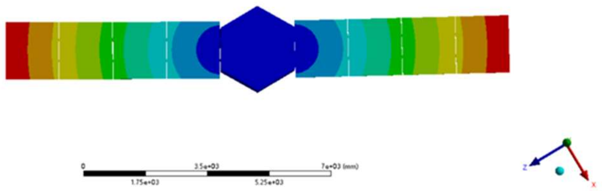

Figure 3. Results of static analysis

When the target star body model undergoes spinning motion around the ontology with a maximum angular acceleration of 1.13 rad/s2 (spin axis parallel to the normal of the solar sail), the stress is shown in Figure 4. The point of maximum stress is located at the connection of the solar panel, with a maximum stress value of approximately 59.349 MPa. The maximum stress value is lower than the safe strength of the hinge connection at the root, and the stress values at other locations are below the allowable stress for the material. Therefore, the structural strength meets the requirements of the technical specifications.

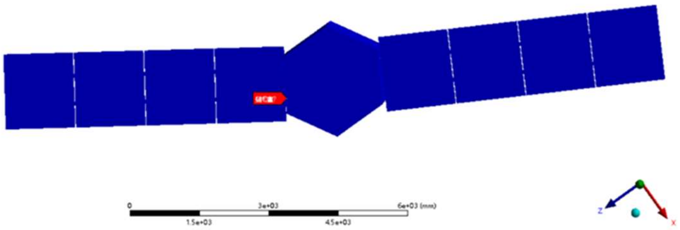

Figure 4. Analysis Results of Rotational Acceleration

When the target star body is subjected to collisions during transportation and handling, the forces act on the outermost points of the solar sail. The direction of the force is along the edge of the solar sail, pointing from the front side of the target star to the rear side. The magnitude of the force is 150N. Calculate the deformation as shown in Figure 5. From the deformation chart, it can be observed that the solar sail of the target star undergoes a certain amount of deformation. The maximum deformation occurs at the outermost point of the solar sail, with a deformation of approximately 0.02mm. The entire structure of the target star is intact, indicating that the ontology possesses sufficient rigidity.

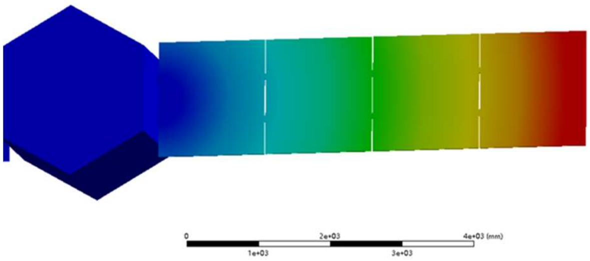

Figure 5. Analysis Results of Collision Force

4. Modal Analysis

Modal analysis is a method used to determine the mode shapes and modal vectors of a system with a finite number of degrees of freedom, under the assumption that the system is in a steady state with stable loading, temperature, and pressure conditions. It involves solving the equations of motion for undamped and load-free states. The matrix expression for the undamped free vibration equation is:

$$[M]\{\ddot{u}\}+[K]\{u\}=0 \tag{1}$$

In equation (1), [M] represents the mass matrix of the structure, [K] represents the stiffness matrix of the structure, {u} represents the displacement vector, $\{\ddot{u}\}$: acceleration matrix vector.

When there is damping, the control equation for the target star under the absence of external forces is:

$$[M]\{\ddot{u}\}+[C]\{\dot{u}\}+[K]\{u\}=0 \tag{2}$$

In equation (2), [M] represents the damping matrix, $\{\dot{u}\}$: velocity vector.

Let its solution be(3):

$$\{x\}=\{\psi\}e^{\lambda t} \tag{3}$$

Substituting into equation (2):

$$(\lambda^2[M]+\lambda[C]+[K])\{\psi\}=[D(\lambda)]\{\psi\}=0 \tag{4}$$

In equation (4), $[D(\lambda)]$ is the characteristic matrix of the system.

Requirements for the target star mode are the fundamental criteria for structural design. Since modal analysis depends on the stiffness of the structure, the requirements for the target star mode are sometimes referred to as stiffness requirements for the structure. Since modal analysis depends on the stiffness of the structure, the requirements for the target star mode are sometimes referred to as stiffness requirements for the structure. Performing modal analysis on the designed target star structure allows us to determine if its frequencies meet the requirements. The target star can be divided into two parts: the overall star and the star body. The entire star includes the two side solar panels, while the body of the star does not include the two side solar panels. Calculate the frequencies of the first six modes.

First, modal analysis was performed on the entire star. In Figure 6, it can be seen that the first mode result is 0.774Hz, which exceeds 0.2Hz. The flexible attachment meets the requirement to have a frequency not less than 0.2Hz. Modal analysis of the star body reveals that the first mode result is 20.36Hz, as shown in Figure 7, which meets the design requirement of being at least 20Hz. It is observed that without the two side solar panels, the star body experiences excessive vibration at higher frequency ranges. In the case where the overall star includes the two side solar panels, the main source of vibration is the vibrations occurring in the two side solar panels, with lower frequencies of vibration.

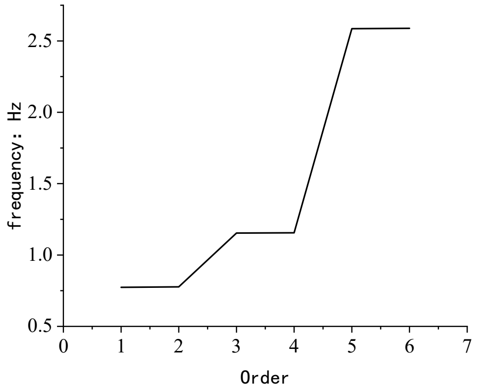

Figure 6. Frequency response curve of the overall star.

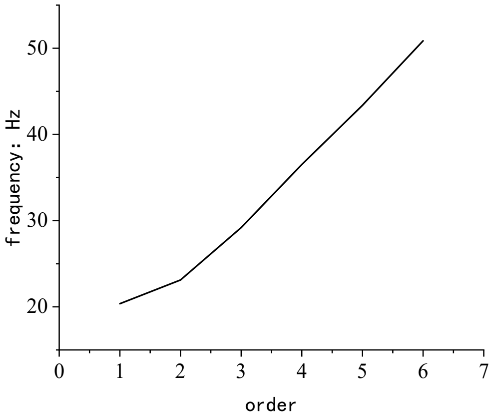

Figure 7. Frequency response curve of the star body.

5. Conclusion

(1) Finite element simulation analysis was conducted on the target star structure, including static analysis, rotational acceleration analysis, collision force analysis, and modal analysis. Based on the calculation results, it can be concluded that the designed star body structure meets the requirements.

(2) When the target star body model undergoes spinning motion around its body at the maximum angular velocity, the point with the maximum stress is located at the connection of the solar panels. The maximum stress value is approximately 59.349MPa, with a safety margin of 2.5, indicating good stability.

(3) During high-speed rotation of the target star, the influence of the atmosphere on the star body was not considered. In the presence of air, rotation would generate air resistance, causing deformation and changes in stress within the star body structure. In future research, it is necessary to study this specific operating condition and its impact.

References

[1] Gu Dequ, Cen Junhan, Chen Jiajia. Satellite Communication Antenna Structre Design Method and Model Design Analysis Based on 5G Background. 2021 International Wireless Communications and Mobile Computing, (Virtual, Online, China, November34-36, 2021). Vol.6, p.610-614.

[2] Li Hang, Liu Ruiyao, Xin Haijun, et al.Ant-Inspired Bionic Design Method for the Support Structure of the Fengyun-3 Satellite Payload Infilled with Lattice Structure. Materials, Vol.16. (2023)No.2, p.16-23.

[3] Xu Xin, Liu Zhijia, Hu Yongqiang, et al. The Structrue Stiffness Optimization Design and Test of a Satellite. 2022 5th World Conference on Mechanical Engineering and Intelligent Manufacturing, (Virtual, Online, China,November18-20, 2022). Vol.6, p.319-325.

[4] Tao Xiao, Ding Wei, Zhang Xingang, et al. Multiaperture Multibeam Antenna Design Method for High-thoughput Satellite Applications. nternational Journal of RF and Microwave Computer-Aided Engineering, Vol. 32(2022)No.6, p.234-248.

[5] Singh Vimlesh, Kumar Amit, Aghwariya, et al. Design and Analysis of DGS-Based Fractal Antenna for Metrological Satellite. Springer Science and Business Media Deutschland GmbH, Vol25(2022)No.7, p.247-255.

[6] Mindarno H S, Huzain M F, Setiawan M N. Design Structure and Integrity Simulation of SURYA Satellite-1 Structure Using Ansys Workbench. Proceedings of the International Astronautical Congress, IAC(Bremen, Germany, November1-5, 2018)No.8, p.75-84.

[7] Yang Jiandong, Li renlong, Ni Xiao guang. Design of Separating Structure of Small Satellite with Elastic Metal. Proceedings - 2020 2nd International Conference on Artificial Intelligence and Advanced Manufacture, AIAM(Shanghai, China, November1-2, 2020)No.26, p.412-415.

[8] Rianto Puji, Herawan Agus, Kurniawan Rendy. Development of remote sensing satellite attitude visualization simulator: Mechanical design. Proceedings of SPIE - The International Society for Optical Engineering, Vol11372(2019)No.8, p.235-242.

[9] He Junwei, Zhang Yadong, Shangguan Zeyin. A Review of Bionic Design in Satellite Solar Wing Structures. Journal of Physics: Conference Series, Vol1549(2020)No.7, p456-463.

[10] Wang Yan, Liu Rongqiang, Yang Huli. Optimization design and modal analysis of satellite SAR deployable support structure. Applied Mechanics and Materials, Vol635(2014)No.9, p185-189.