Aerospace and Electronics (2021) 2: 9-18
Clausius Scientific Press, Canada
DOI: 10.23977/aeroe.2021.020102
ISSN 2523-5907

Analysis and Optimization of the Operating Environment of a Star Sensor

Junfa Duan, Xi Yang*

School of Mechanical Engineering, North China University of Water Resources and Electric Power, Zhengzhou, 450003, China
**corresponding author*

Keywords: Star tracker; mechanical structure; finite element analysis; parameter optimization

Abstract: Navigation star tracker is an important part of spacecraft to identify its own position and attitude in space. Its mechanical structure must be able to ensure that it will not produce large stress when it is subjected to vibration and impact during the launch process, which will damage the detection accuracy and imaging quality of the navigation star tracker. At the same time, the spacecraft design is very sensitive to the mass requirements, so the navigation star tracker must be designed to minimize the mass while ensuring the structural strength. According to the design and application requirements of a star tracker, the working conditions of the preliminary design scheme are analyzed by using workbench, and the optimization is carried out according to the analysis results. Under the condition of meeting the service conditions, the optimization results can reduce the structure weight and provide reference for the design of subsequent series.

1 Introduction

Navigation star sensor is commonly used in spacecraft to determine its azimuth and attitude. Generally, the hardware part of star sensor is composed of optical lens group, lens barrel spacer and other mechanical parts, hood and supporting circuit. In order to observe different star-point targets and ensure high imaging quality, the star sensor requires a relatively complex and high-precision hardware structure, especially the mechanical parts such as the lens barrel spacer that cooperates with the optical lens group [1-3].

During the spacecraft's lift-off process, various components will be subjected to loads such as random vibrations and shocks, which will have varying degrees of impact on the internal mechanical structure. Because of its special complex and precise mechanical structure, the star sensor is highly sensitive to loads such as vibration and impact. High-frequency loads are prone to resonate with the mechanical structure to generate large local stresses, which affect the accuracy of the structure and even cause component collapse. Cause damage to the sensor. Therefore, it is necessary to simulate and analyze the structure model during the design process to avoid problems such as resonance and damage to the sensor due to large stress. At the same time, due to the particularity of the spacecraft, in order to control its launch cost, it is necessary to reduce the quality of the preparation as much as possible and improve the quality of the load. Therefore, in the design process of each component, parameter optimization must be carried out under the premise of meeting the function and strength, and the quality of each component must be reduced as much as possible[3,4].

In this paper, modal analysis, sinusoidal sweep analysis, random vibration analysis and impact load analysis are carried out for a certain type of navigation star sensor. Under the condition that the model meets the strength requirements, the structure size and material are optimized. By comparing the optimization results, the design is improved and the design scheme is finally determined [4].

2 Structural Design and Analysis

2.1 Physical Design

In this paper, the collimation objective part of the navigation star sensor is analyzed and optimized. Based on the calculation results of Code V11.0, the mechanical structure of the collimation objective of the sensor is preliminarily designed.

As shown in Figure 1, the structure is composed of five lenses, a primary lens barrel, a sub-lens barrel, and two pressure rings and three spacers inside. The star sensor is connected to the fixing base by the Φ62 flange on the primary mirror cylinder, and the small flange on the outside of the sub-mirror cylinder is used to connect the light hood. Each Material properties of components are shown in Table 1.

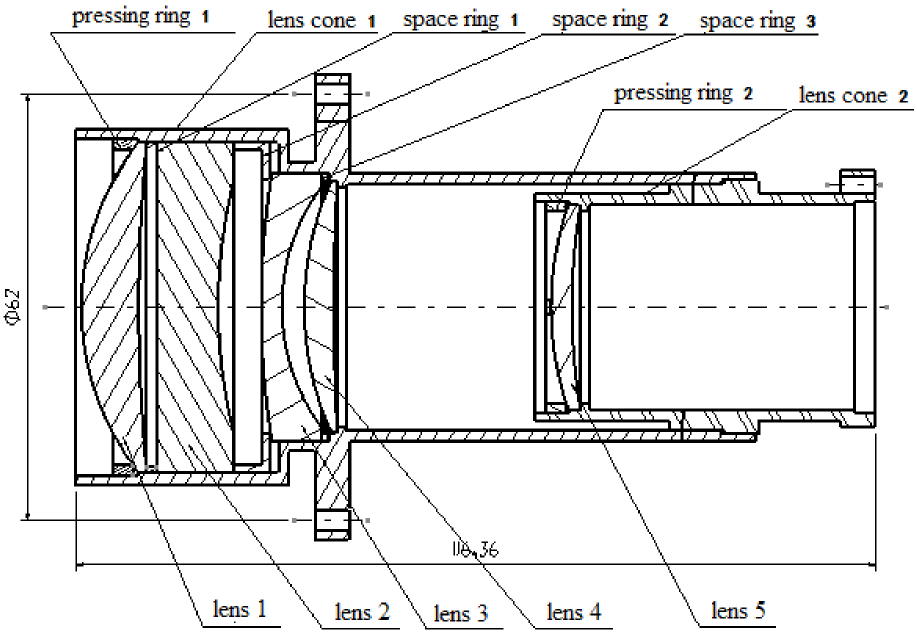


Figure 1: Preliminary design scheme structure.

Table 1: Material parameters.

Name of parts		materials	density Kg/m3	elasticity modulus (Pa)	Poisson's ratio	coefficient of thermal expansion 10^{-7}/K
battery of lenses	lens 1	H-LAF62	3890	1.0143E+11	0.292	70
	lens 2	H-ZF2	2900	8.321E+10	0.241	85

	lens 3	H-ZF11	2950	8.41E+10	0.248	90
	lens 4	H-LAF3B	3780	9.781E+10	0.294	80
	lens 5	H-K9L	2470	6.542E+10	0.22	87
lens cone,; space ring, and pressing ring		4J36	8120	1.34E+11	0.25	1.6
lens cone,; space ring, and pressing ring		TC4	4510	1.1E+11	0.34	9.5
lens cone,; space ring, and pressing ring		6061	2770	7.1E+10	0.33	23
Note: The applicable range of thermal expansion coefficient of all lens materials is -30~+70℃ (the manufacturer does not provide below -30℃); The cylinder material TC4 is a nonlinear material, and the thermal expansion coefficient varies with temperature.						

2.2 Environmental Condition Model Analysis

2.2.1 Experimental environment analysis

Modal analysis is used to extract the dynamic characteristics of the structure, and it is only related to the geometric shape, geometric size, structural materials and constraint mode of the structure. The most important in modal analysis are the natural frequency and mode shape. The natural frequency is the frequency that the structure naturally tends to vibrate when it is disturbed. The deformation tendency of the structure at this frequency is called the mode shape. Through modal analysis, the frequency of resonance and the influence of each frequency on the structure can be understood, and the structure design can be improved to avoid resonance and bad deformation [5,6].

Because the amplitudes of each node in each mode shape are relative, the absolute value can be arbitrary. In practical work, the specific values of mode shapes are usually determined by the normalization and regularization of mode shapes, and only the first six modes are generally analyzed.

Sine scanning test is mainly used for modeling the rockets and spacecraft flight startup and shutdown caused by the engine thrust in the process of change, the transonic air pulse pressure and the rockets through the transonic runtime with attack Angle of impact load, the rockets thermal separation between all levels of the overall structure of the internal release caused transient response, and low frequency vibration caused by longitudinal motion. The frequency is generally below 100Hz [7]. Sine scanning test is generally the test piece placed in the test bed, the test bed in accordance with a certain scanning rate, from low frequency vibration gradually transition to high frequency vibration, test whether the test piece damage. The vibration frequency in the test environment is time dependent. However, in the simulation environment, if the finite element transient analysis is used to simulate the sinusoidal sweep test, the time of a cycle test is long and the number of mesh nodes in the model is large, which requires a large amount of computational resources. In addition, the highest frequency is 100Hz. In order to ensure the accuracy of analysis, the time step must be set very small to further increase the amount of calculation. Therefore, the finite element frequency response analysis is generally used to simulate the sinusoidal sweep test. The principle of frequency response analysis is that the software discretize the frequencies into various frequency points according to the set frequency range and step size, and then load the corresponding load amplitude of each frequency point to the structure in turn, so as to obtain the steady-state results of the structure at each frequency point, without introducing time variable in the calculation.

Frequency response analysis takes the frequency and amplitude of the input signals as variables to calculate the dynamic response of the structure under the action of steady vibration excitation. After calculating the system response in the modal coordinates, the modal response is transformed into the physical response.

The stochastic vibration analysis is used to calculate the state of the structure under the excitation which cannot be defined in time. The result is usually the standard deviation of the structure under a certain response.

The impact spectrum response analysis adopts the method of modal merging. Firstly, the participation factors of each mode of the structure to the excitation direction are calculated, then the maximum response under the mode is calculated, finally, the maximum response of each mode is combined in some way to get the total response of the structure[8].

2.2.2 Experimental conditions

The experimental conditions required for the experiment are shown in Table 2 below.

Table 2: Sine sweep vibration test conditions.

project	frequency domain（Hz）	qualification test
Axis, normal and horizontal directions	5	0.6g
	50	3.2g
scan rate（oct/min）	24	
number of experiment	Each time	
function test	Power on test under shock vibration, each function should meet the design requirements	

The random vibration test conditions are shown in Table 3.

Table 3: Random vibration test conditions.

frequency domain（Hz）	qualification test	
	power spectral density	The total mean square value
20~80	3db/oct	6.06g
80~350	0.04g2/Hz	
350~2000	-3db/oct	
direction of vibration	Three directions	
Action time	Defect elimination phase 3min/ direction, no fault inspection phase 3min/ direction.	
connecting format	The test product is rigidly connected with the shaking table.	
Vibration control point	It is located on the test product and connected with the	

	shaking table.
detection mode	The whole machine is energized during vibration monitoring.
special circumstances	a) If three directions of vibration cannot be realized, two directions of vibration can be performed according to the actual loading direction and the sensitivity of the failure mode to the direction, and each direction is 7min. b) When the whole machine is screened with vibration, attention should be paid to the protection of moving parts.

The impact test conditions are shown in Table 4.

Table 4: Impact test conditions.

frequency domain（Hz）	Identification test, acceleration shock response spectrum
50	150g
1000	5000g
5000	5000g
Impact times	Each time
Function test	Power on test under shock vibration, each function should meet the design requirements.

3 Stability Analysis

In order to ensure the uniformity of the model with other mechanical structure materials, 4J36 and other mechanical parts such as mirror cylinder spacer were used in the initial design. The model was imported into the Workbench and the end face of the large-end flange was fully constrained to analyze the main mode modes of the first six order of the structure. The analysis results are shown in Figure 2.The natural frequency of the first main mode is 2616.3Hz, followed by 2620.7Hz, 4419.1Hz, 4441Hz, 5867Hz and 5872.5Hz.Under the first vibration type, the star sensitive head swings along the X direction, the second vibration type swings along the Z direction, the third vibration type stretches along the Z direction, the fourth vibration type stretches along the X direction, the fifth vibration type expands, twists and shrinks, and the sixth vibration type expands, twists and shrinks. The deformation trend of the structure is relatively symmetrical under different modes, and there is no sensitive part to be strengthened and optimized locally, and the natural frequency of the main mode of the structure is high.

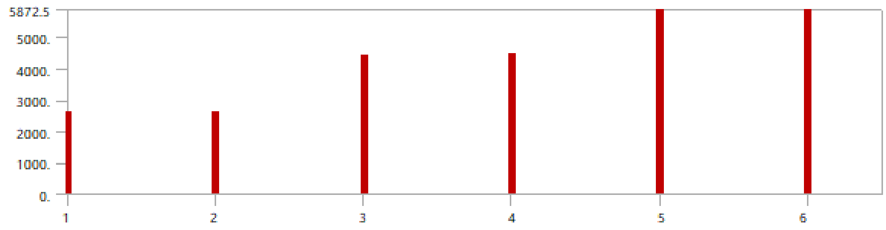


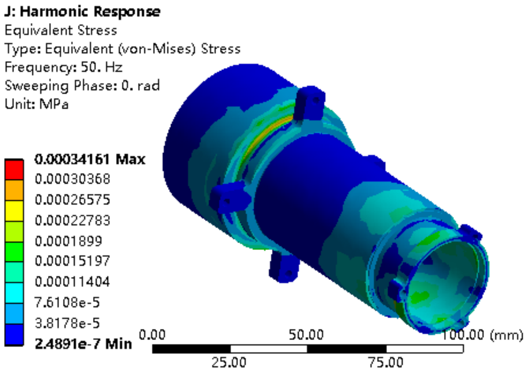
Figure 2: Natural frequency/Hz of the first six modes.

The sinusoidal scanning analysis results of the model are shown in Figure 3. The maximum axial sinusoidal scanning stress is 3.416E-4MPa, and the maximum radial stress is 0.168MPa. The maximum stress appears at the root of the end face of the fixed flange.Because the sinusoidal scanning frequency is much lower than the natural frequency of the first main mode, the structural strength is sufficient.

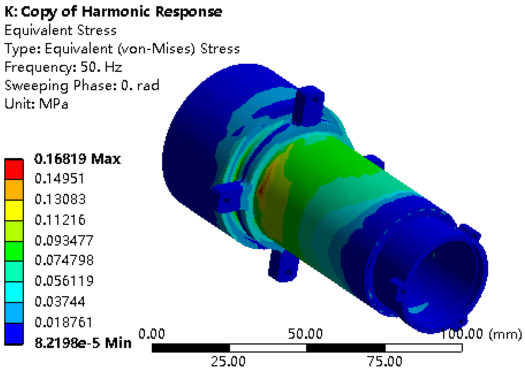
Random vibration analysis requires conversion of power spectral density into units required by the software. The conversion formula is as follows:

$$m(db/oct) = 10 * lg(A2/A1)/log2(f2/f1) \tag{1}$$

The unit of power spectral density after conversion is $[\left(\frac{\mathrm{mm}}{s^2}\right)^2 \mathrm{Hz}]$, both axial and radial acceleration loads were loaded at the same time, and the simulation results were shown in Figure 4.The results of random vibration analysis show that the maximum stress is 0.399MPa, and the maximum stress point occurs near the root of the fixed flange with a probability of 68.269%. The structural strength is sufficient.



a. Axial sinusoidal scanning stress cloud.

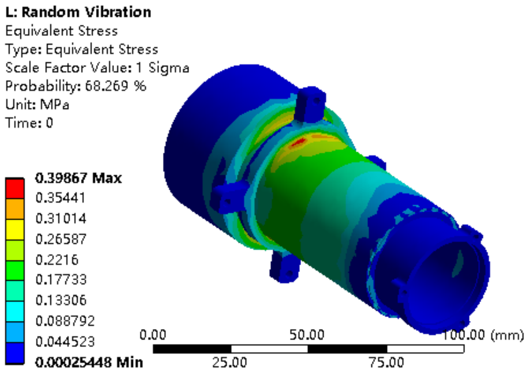


b. Axial sinusoidal scanning stress cloud.

Figure 3: Sine sweep result.

	Frequency [Hz]	Acceleration [(mm/s²)²/Hz]
1	20.	162.4
2	100.	200.
3	1300.	200.
4	2000.	2.032

a Power spectral density.



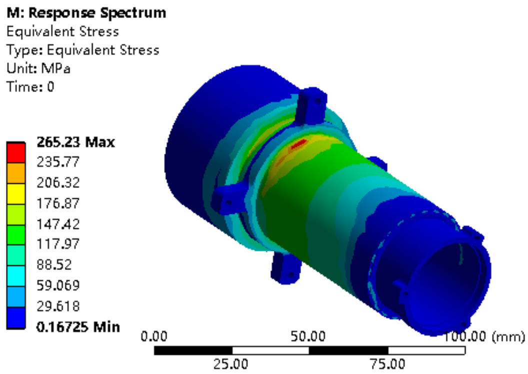
b Random vibration stress cloud diagram.

Figure 4: Random vibration results.

Frequency domain acceleration curve and analysis results of impact load analysis of the model are shown in Figure 5.The maximum stress of the structure is 265.23MPa, and the maximum stress point is near the root of the fixed flange, which is less than the maximum allowable stress of 4J36 material, 500MPa, and the structure has sufficient strength.

	Frequency [Hz]	Acceleration [(mm/s²)]
1	50.	132.67
2	800.	8000.
3	5000.	8000.

a Frequency domain accelerometer.



b Impact load stress cloud diagram.

Figure 5: Impact load results.

In summary, the structural strength of the initial design meets the use demand, and the total weight of the structure is 593g.Since the user requires that the total design weight should not exceed 500g, the initial design needs to be further optimized.

4 Comparison of Optimization Results

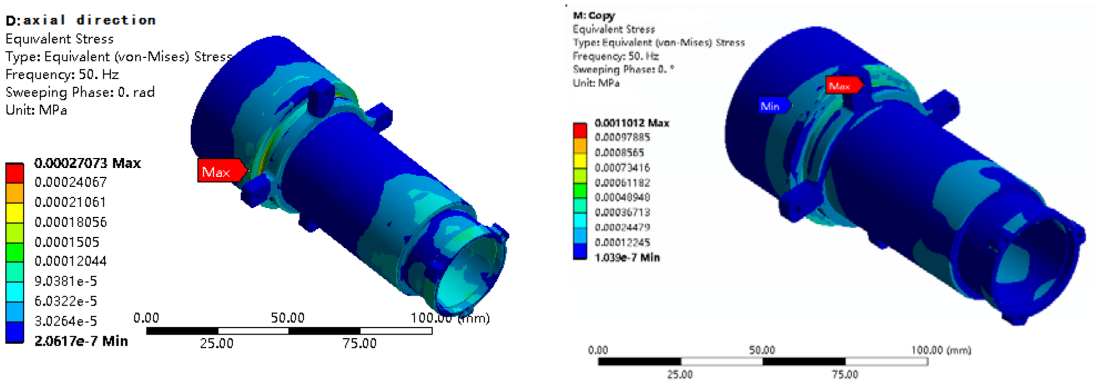
The optimal design of the initial scheme is divided into two optimization directions: structure size and structure material .Scheme 1: Replace the mechanical parts with smaller density titanium alloy TC4 to reduce the total weight; In the second scheme, the total weight is reduced by optimizing the structure and reducing the size. Only the wall thickness of the original two-stage mirror cylinder is reduced from 2.5mm to 1.5mm, and 4J36 Yintile alloy is still used to ensure the unity of the structural materials assembled with it.

The modal analysis results show that the variation trend of the two is the same as that of the initial scheme, and the natural vibration frequencies are shown in Table 5.After analysis, the natural frequencies of the two optimization schemes are higher than those of the original scheme.

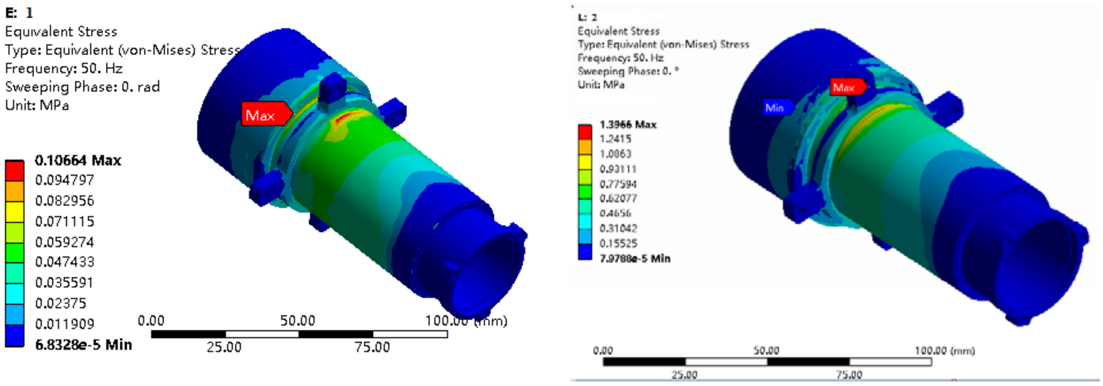
Table 5: Natural vibration frequency of optimization scheme.

	1/Hz	2/Hz	3/Hz	4/Hz	5/Hz	6/Hz
Plan a	3317.9	3324.1	4789.5	4811.4	7294.8	7581.3
Plan b	4218.2	4220.3	5793.3	5935.7	7858.1	7858.7

The sinusoidal scan results are shown in Figure 6.The maximum axial and radial stresses of the optimal scheme 1 are 2.707E-4MPa and 0.107MPa, respectively. The maximum axial and radial stresses of the second optimization scheme are 1.1E-3MPa and 1.397MPa, respectively. In the sinusoidal scanning simulation, the maximum stress of Scheme 1 is not different from that of the original scheme, while the maximum stress of Scheme 2 increases by about ten times. Therefore, in the sinusoidal scanning analysis, the influence of the contour size of the model is greater than that of the model material.



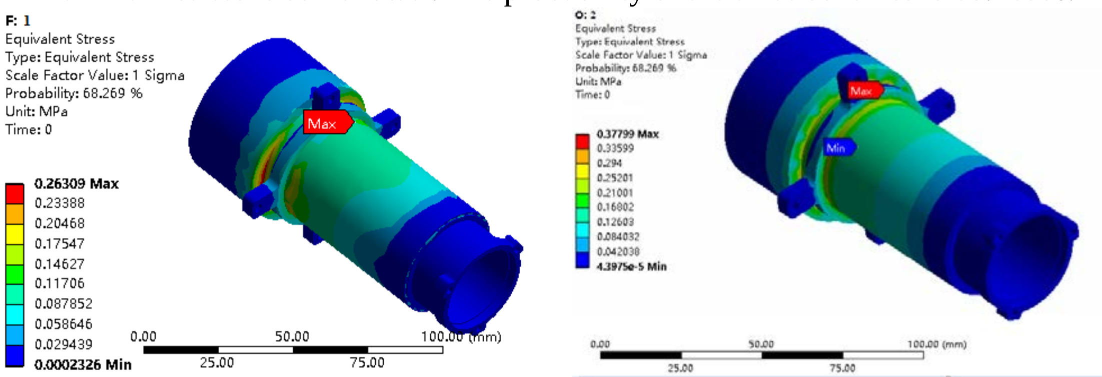
a Plan 1 axis b Plan 2 axis



c Plan 1 radial d Plan 2 radial

Figure 6: Forward scanning stress nephogram.

The random vibration analysis results of the two optimization schemes are shown in Figure 7.The maximum stress of scheme 1 is 0.263MPa, and the maximum stress of scheme 2 is 0.378MPa.Plan the maximum stress of two with the initial 0.399 MPa is relatively close, so the random vibration analysis, the material is bigger, the influence of the results under the condition of invariable in shape size 4j36 replacement for TC4 after the initial material structure maximum stress significantly decreased, and decrease the mirror wall thickness is not only change the material of scheme 2 maximum stress is some lower. The probability of the three schemes is 68.269%.

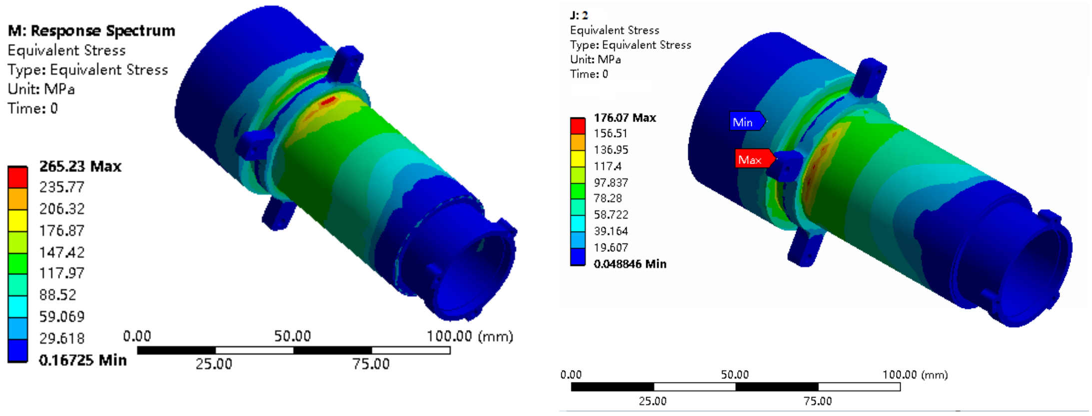


a. Plan 1 Stress nephogram. b. Plan 2 Stress nephogram.

Figure 7: Random vibration stress nephogram.

The analysis results of impact load model are shown in Figure 8.The maximum stress of the initial design structure is 265.23MPa, the maximum stress of the first optimization scheme is

265.23MPa, and the maximum stress of the second optimization scheme is 176.07MPa. Therefore, the analysis results of the impact load of this type of structure are more sensitive to the change of the structure size. With the decrease of the wall thickness of the mirror cylinder, the mirror wall quality decreases. The maximum stress decreases under the same impact load.



a. Plan 1 Stress nephogram.

b. Plan 2 Stress nephogram.

Figure 8: Impact load stress nephogram.

According to the above analysis and comparison, it can be concluded that sinusoidal scanning and impact load are more sensitive to the size (mass distribution) change of the structure under two experimental conditions, while random vibration load is more sensitive to the change of model material. The total weight of the structure in the first optimization scheme is 362g, and the total weight in the second optimization scheme is 320g.By replacing the material with a lower density, the total weight can be greatly reduced without changing the initial design while ensuring the strength. However, considering the coordination with other components, if the material is not unified, it is easy to lead to stress concentration during the thermodynamic cycle test, which brings inconvenience to the optimization design. Reducing the wall thickness of the mirror barrel and other components through optimization design can also reduce the total weight while ensuring the strength. However, due to the limitation of processing conditions and processing costs, the wall thickness of the components cannot be reduced unlimitedly. Therefore, we choose to reduce the design wall thickness from 2.5mm to 1.5mm under the condition of matching the current processing conditions. Based on the above comparison, the second optimization scheme is selected as the final design scheme.

5 Conclusion

In this paper, the initial design is carried out according to the design requirements, and the initial design model is imported into the analysis software for modal analysis, sinusoidal sweep analysis, random vibration analysis and impact load analysis. By comparing the results of the two optimization schemes and observing the variation trend under each test, a better optimization scheme was finally selected, which provided design ideas and optimization direction for the subsequent structure optimization and product iteration.

References

[1] Wu, Y. X.; Qiao, J.; Wang, L. P.; Optical system design of long focal length non-thermal star sensor. Infrared and Laser Engineering, 1-14(2020-08-09).

[2] Qian, Z. Y.; Luo, W. B.; Yin. Y. Z.; Zhang. Ling, B. G.; Cai, Z.; Fu, W. C.; Lu, Q. R.; Zhang, X. W.; Zhao, C. G.; Design and Verification of the Dimensional Stability of Gaofen-7 Satellite Structure. Chinese Space Science and Technology, 1-8(2020-08-09).

[3] Zhang, H.; Zhou, X. D.; Wang, X. M.; Tian, H.; Progress and Development of All-Sky Star Sensor in Near Earth Space. ACTA AERONAUTICAET ASTRONAUTICA SINICA, 1-14(2020-08-09).

[4] Meng, X. Y.; Research on Design and Installation Technology of Star Sensor Lens. Changchun University of Science and Technology. (2019).

[5] Wang, Z. G.; Zhang, Z. H.; Chen. S. Q.; Mao, M.; Spacecraft Engineering. (2019), 28(03), 56-63. (In Chinese).

[6] Fang, X.; Bi, J. D.; Analysis on the Variation Phenomenon of System Characteristics in Vibration Environment. Chinese Congress of Mechanics (CCTAM 2019).

[7] Yin, Z.; Hou, X. Y.; Guo, J. H.; Gao, H. Y.; Spacecraft Environmental Engineering. (2017), 34(02), 214-221.

[8] Shen, B.; Research on Mechanical Environment Numerical Simulation and Reliability Design of Arrow-borne Electronic Equipment. China Academy of Launch Vehicle Technology. (2019).