Computer intelligent dynamic simulation and optimization for structural platform of water treatment subsystem on the spacecraft

Xiaoxin Wang1, *, Tianfeng Sun1 and Feifei Jiao2

1Beijing Space Crafts Manufacturing Co., Ltd, Beijing, China
2Astronaut Center of China, Beijing, China

*Corresponding author: xiaoxinwang@cast.cn

Abstract. In order to reduce the load response condition of spaceborne equipment, an optimized design scheme of carbon fiber composites sandwich panel was proposed in this paper. The state before and after optimization was analyzed and compared through simulation calculation. The comparison shows that after the honeycomb sandwich structure was optimized, its fundamental frequency increased by 75%, the drawing force of embedded parts decreased by 50%, and the maximum acceleration response also decreased. Finally, the response under sinusoidal vibration was evaluated by test, and the results show that the optimization of the structure is reasonable and feasible, it can be the reference of other similar products.

Keywords: honeycomb sandwich composite, finite element method, fundamental frequency, rigidity.

1. Introduction

The honeycomb sandwich structure is light and has excellent performance, which has large bending stiffness and small bending stress under the same weight[1], and has excellent fatigue resistance, sound absorption and vibration reduction performance [2],[3], which is widely used in various bearing structures. The performance characteristics of the cellular sandwich structure determine its application in the aerospace field, and is an important part of the various aircraft and spacecraft structure.

Most of the research on equivalent parameters in honeycomb sandwich surface was developed according to the cellular material theory proposed by Gibson [4]. Gibson gave the more comprehensive equivalent parameter calculation formula, based on which the complex honeycomb sandwich plate structure can be analyzed equivalent to laminboard structure. However, the theory also has its obvious shortcomings. Based on this, later generations, many new computing models have been developed. These models can be roughly attributed to three categories of: equilibrium models, displacement models and hybrid models, and the application range of various models varies greatly.

At present, the cellular sandwich structure is widely used in the space field, mostly installed by spacecraft instruments. To reduce the load response conditions of the instrument equipment, the instrument installation board shall also meet the stiffness and strength requirements while providing the installation interface.
2. Characteristics of structure

A structural platform of water treatment subsystem is the main bearing structure of each single machine, pipeline and cable of the spacecraft subsystem. It is composed of the bottom plate and the neutral plate, which is connected to the spacecraft substructure through the structure connection hole. Its outer envelope is defined and requires functional and performance requirements at a limited weight, as shown in Fig. 1.

A honeycomb sandwich panel design scheme with high-mode carbon fiber composite material and aluminum honeycomb core is proposed in this paper. The carbon fiber composite panel is M55J/epoxy based composite, preimmersion thickness is 0.1mm, paving sequence: bottom plate paving sequence is [0 / 45 / -45 / 90] s, total thickness of 0.8mm; The neutral plate paving order is [0 / 45 / -45 / 90], and total thickness of 0.4mm, honeycomb cell is 0.05mm × 3mm.

In order to improve the structural base frequency and local stiffness and reduce the maximum response of the equipment installation area. This study optimized the honeycomb structure of the sandwich.

3. Design optimization

According to the preliminary modal analysis results, three strengthening beams are embedded in the honeycomb core of the bottom plate, and the embedded parts are embedded in the beam into the beam through the structural connection hole constraint. The water tank should be installed at the large circular open hole, where the panel is discontinuous and the strength and stiffness are weak. In order to improve the strength of the water tank installation point, the embedded beam C(ring beam is embedded in the honeycomb core of the area), and the finite element model is shown in Fig. 2.

In order to improve the local stiffness of the structure, the optimized scheme added three embedded straight beams with cross sections of C and H, with a total of six embedded straight beams inlaid. In addition, according to the local response analysis, three inclined carbon fiber support rods were added between the neutral plate and the bottom plate, connected to the beam inside the plate.
Figure 2. Finite element model of the installation board

The position of the embedded beam is shown in Fig. 3. The materials of the embedded beam are M55J/ epoxy 648 one-directional preimmersion material, and the single layer thickness is 0.1mm, 2mm thick beam paving direction is [0 / +45 / -45 / 90 / 0] S2, fiber direction of 0° is the length direction.

Figure 3. Diagrammatic sketch of beam reinforcement

Parameters of aluminum honeycomb equivalent materials and material properties of M55J/ epoxy composite are shown in Table 1-Table 2.

| Table 1. Equivalent mechanical properties of honeycomb and high modulus carbon fiber/epoxy unidirectional lamina |
|---------------------------------------------------------------|
| **honeycomb** | $E_{11}$/GPa | $E_{22}$/GPa | $G_{12}$/MPa | $G_{23}$/MPa | $G_{13}$/MPa | $\rho$/ (kg/m$^3$) |
| honeycomb     | $0.75 \times 10^3$ | $0.75 \times 10^3$ | 0.3   | 10 | 156 | 234 | 68 |
| unidirectional lamina | 6.8 | 6.8 | 0.28 | 7200 | 7200 | 7200 | 1700 |

Simplification of finite element model:

a. The bottom plate and neutral plate are simplified to composite housing units with a unit size of 5mm, bottom plate connected to the neutral plate through a rigid MPC unit;
b. Embedded square and C-shaped beams are reduced to beam units;
c. Instrument: main equipment on each structural board are simplified to centralized quality element, at the center of mass and given rotational inertia attribute. The centralized quality unit is connected to the structural board through rigid MPC unit and hole of the equipment; other quality such as pipeline, cable and fasteners are applied to the model through non-structural quality.
3.1. Modal analysis

Using the finite element model, the pattern of the vibration of the structure plate is shown in Fig. 4. It is seen from the calculation results that the base frequency of the structure is 66.0Hz.

Figure 4. Mode shape of the installation board before optimization

The optimized scheme model is calculated to get the first fourth order vibration type of the installation plate as shown in Fig. 5. The results show that the base frequency of the mounting board is 114.2Hz, with a 73% increase and greater stiffness compared to the pre-optimized 66.0Hz.
3.2. **Strength analysis**

The maximum tensile strength of the embedded parts under the worst load condition (Y to 15g) is obtained, as shown in Table 2. Point 1 is the structural connection point of the installation plate and the cabinet, its support force as shown in the table, point 2 is the connection point of the single equipment and the installation plate, and point 3 is the connection of the installation plate and the neutral plate. It can be seen that the design optimization effect is obvious, and the pulling force of different positions is lower than the allowable pulling force of the embedded parts, which meets the strength requirements.

| Connections | Force before optimization /N | Force after optimization /N | Allowable force /N |
|-------------|-----------------------------|---------------------------|-------------------|
| No.1        | 2090                        | 822                       | 2000              |
| No.2        | 1460                        | 585                       | 2000              |
| No.3        | 2830                        | 1410                      | 2000              |

3.3. **Frequency analysis of equipment**

Using the identification sine vibration load condition, the acceleration response of the structural plate and individual equipment is analyzed in three directions (X, Y, Z). The damping coefficient of the structure is taken uniformly as, and the boundary conditions are the simple support constraints at the overall connection point of the bottom plate and the neutral plate and the cabinet.

FIG. 6 shows the maximum acceleration response of each single unit during X, Y, Z-direction excitation. At X excitation, the instrument and equipment have the maximum acceleration response at 60.0Hz with a size of 6.7g, and is shown in Fig.6(a).

During the Y direction excitation, the acceleration frequency sound curve of the instrument and equipment is shown in Fig. 6(b), and there are 5 single machines with the acceleration response exceeding 15g.

During Z direction excitation, the instrument and equipment have the maximum acceleration response at 66.0Hz frequency and the 9.7g, acceleration frequency sound curve is shown in Fig. 6 (c).
Figure 6. Acceleration frequency response curves of instruments

With the weakest bending stiffness of the bottom plate Y in the outward direction and the largest response under the Y sinusoidal vibration, the Y acceleration response of the optimized bottom plate is analyzed. Fig. 7 shows a maximum acceleration response cloud map of the mounting plate at different frequency points Y to the identification level sine excitation conditions. It is seen that the bottom plate responds most at 100Hz frequency point Y towards sinusoidal excitation with a maximum reduction of 7.25g, compared to prior optimization.

The bending stiffness of the neutral plate Z is the weakest, so the optimized Z direction acceleration response is analyzed. Fig. 8 shows a maximum acceleration response cloud map of the mounting plate at different frequency points Z to the identification level sine excitation conditions. It is seen that the bottom plate responds most at 100Hz frequency point Z towards sinusoidal excitation with a maximum decrease of 6.2g, compared to pre-optimization.
From the above simulation results, the maximum response position is in the middle position on both sides of the bottom plate, and the acceleration curve of the maximum response position in the region is extracted as shown in Fig.9.

Figure 7. Acceleration response nephogram of the installation board in Y-direction excitation

Figure 8. Acceleration response nephogram of the installation board in Z-direction excitation

Figure 9. Acceleration frequency response curves of maximum response position after optimization
4. Test and verification
In order to verify the design scheme, we conducted the vibration environment test after the optimized design, analyzed the data of the acceleration response measurement point in the vibration test, and drew the response curve of each acceleration measurement point.

During the vibration test, the vibration control adopts a four-point average method. The control point is set on the connection surface of the connecting board and the tooling and the four corners of the product using 20t vibration platform system. Installation during the vibration test is shown in FIG.10.

![Figure 10. Installation photograph of vibration test](image)

The response curve of the appraisal sine vibration test is shown in Fig.11. It is seen that the test data and the design simulation curve fit high, no characteristic frequency drift before and after the vibration test, and no structural damage after the vibration test, indicating that the structural scheme has been verified and the design is reasonable and feasible.

![Figure 11. Y-direction acceleration frequency response curves of qualification test](image)

5. Conclusion
A honeycomb sandwich structure of carbon fiber composite skin is proposed in this paper, optimizes the analysis, analyzes the simulation results, and passes the test verification. Through the optimized design analysis of the following conclusions:

(1) The base frequency of the installation board by increasing the embedded beam and inclined support rod is 114.2Hz, which improves about 73% compared with the 66.0Hz before the optimization;

(2) After optimization, the maximum pulling force of embedded parts under the worst load working conditions can be reduced by nearly 50%, and the optimization effect is obvious and meets the strength requirements;
(3) Under the Y-level sine vibration condition, the maximum frequency noise of the optimized installation plate bottom plate equipment is 7.25g, meeting the maximum acceleration response requirements of the equipment;

(4) By comparing the test response curve and the simulation response curve, the two are well matched. The structural design scheme is reasonable and feasible, and can be used as the design reference of similar structures.

References

[1] Vinson J R. The behavior of shells composed of isotropic and composite materials[M]. London: Kluwer Academic Publishers. 1993.

[2] Niu M C Y. Airframe stress analysis and sizing[M]. 2nd Edition. Hong Kong: Hong Kong Commilit Press LTD, 1999.

[3] T Liu. Progress in the vibration control technology of the spacecraft [J]. Journal of Astronautics, 2008, 29 (1): 1-12.

[4] L. J. Gibson, M. F. Ashby. Cellular Solids Structures and properties – Second edition [M]. 1999.