Mechanical Simulation Analysis of Aerospace High Reliability Electronic Equipment

Shang Jiang*, Hongjin Liu, Zucheng Gu, Qun Liu
Beijing SunWise Space Telemetry Ltd, Beijing, 100086, China
*Corresponding author's e-mail: jiangshang_shang@sunwisespace.com

Abstract: In order to meet the requirements of high reliability, the structural performance of aerospace electronic equipment must meet the requirements of long-term stable work under severe mechanical conditions. The finite element model of electronic equipment from structural frame to printed circuit board and important components is established in detail. The mechanical analysis of the whole machine under the working conditions of modal analysis, random vibration and sinusoidal vibration is carried out, and the natural frequency, vibration mode and corresponding stress-strain data of each component are obtained. Through the PCB deformation check and structural strength check, the design safety margin of the whole machine is obtained. The analysis results provide the design basis for the structure finalization of aerospace electronic equipment, shorten the research and development cycle, reduce the experimental cost, and have important significance for the structure design of electronic equipment.

1. Introduction
The use environment of aerospace electronic equipment is more diverse, complex and harsh than other electronic equipment, and its design is more important and difficult. In order to adapt to various harsh environments and work reliably for a long time, the design of aerospace electronic equipment should comprehensively consider various factors affecting equipment performance, including system structure and electrical performance, and take corresponding reliability assurance measures. From the development of the initial sample to the design finalization, the structural response of the strength and stiffness of the equipment under various mechanical environments during its service life should be accurately evaluated to meet the high reliability requirements of aerospace electronic equipment. Generally, simulating the stress response characteristics of actual products through simulation analysis to confirm the rationality of the structure is an effective means to effectively reduce the modification of structural form and the number of tests [1-6].

Based on the environmental test requirements of an aerospace electronic equipment structure, use ANSYS Workbench to establish the mechanical simulation model, set the load and boundary conditions, and carry out modal analysis, random vibration analysis and sinusoidal vibration analysis. Analyze the structural stiffness and strength of the equipment under various working conditions, obtain the corresponding deformation and stress data, and check the safety of the stress-strain data to verify whether the electronic equipment meets the requirements of stiffness and strength, fundamental frequency, etc. The results show that the design of the electronic equipment meets the requirements of high reliability in aerospace.
2. Establishment of structural finite element model

2.1. Establishment of structural finite element model

The electronic equipment adopts modular design and is a vertical cuboid. The adjacent modules of the whole equipment are interconnected through structural lugs, which has good interchangeability. One side plate is connected with the end module at both ends of the equipment to ensure that the whole machine has good electromagnetic compatibility performance. The structure of the whole machine is shown in Figure.1.

Figure 1. 3D schematic diagram of electronic equipment

Considering the efficiency and accuracy of simulation, it is necessary to simplify the model to a certain extent before establishing the finite element model, and remove the smaller holes, chamfers, fillets, etc. in the shell structure. The simplified three-dimensional model is meshed to obtain the finite element model, as shown in Figure.2.

Figure 2. 3D grid model of electronic equipment

2.2. Material parameter setting

Electronic equipment components are mainly composed of structural frame, printed board, connectors and fasteners. The material used for the structural frame structure is aluminum alloy 2A12-H112, with elastic modulus of 70 GPA, Poisson's ratio of 0.3, density of 2800 kg/m3 and yield strength of 275 MPa. The material used for printed board is FR4, with elastic modulus of 13.7 Gpa, Poisson's ratio of 0.14, density of 2800 kg/m3 and yield strength of 338.4 Mpa.

2.3. Boundary conditions

According to the mechanical environment adaptability requirements of aerospace products, the structure needs to bear the mechanical environment effects such as acceleration overload, random vibration and sinusoidal vibration without failure and damage. See Table 1 and Table 2 for mechanical test conditions of qualification level corresponding to aerospace electronic equipment.
Table 1. Random vibration test conditions

| Vibration direction | Frequency range /Hz | Spectral density | Root mean square of acceleration | Time/min |
|---------------------|---------------------|------------------|---------------------------------|----------|
| X, Y, Z             | 20~80 +3 dB/oct     |                  |                                 |          |
|                     | 80~350 0.04 g^2/Hz  |                  | 6 grms                          | 2        |
|                     | 350~2000 -3 dB/oct  |                  |                                 |          |

Table 2. Sinusoidal vibration test conditions

| Vibration direction | Frequency range /Hz | Acceleration/amplitude | Scanning rate / (Oct/min) |
|---------------------|---------------------|-------------------------|---------------------------|
| X, Y, Z             | 10~20 5 mm          | 12g                     | 2                         |
|                     | 20~100 20~100       |                          |                           |

3. Modal analysis

3.1. Modal analysis conditions
Modal analysis is mainly used to determine the natural vibration characteristics of the structure, such as natural frequency, main vibration mode, modal effective mass, etc. Natural frequency and main vibration mode are important parameters in the design of structure bearing dynamic load, and they are also the basis of other kinds of dynamic analysis [7]. In this paper, the natural frequency and formation of structural frame and veneer under the constraint of the whole machine are calculated in ANSYS. The simulation results show that the first 8 modes and vibration modes of the printed board occur in the first mode of the whole machine (the 27th mode of the whole machine), which are obtained under the condition of full constraint of the lug hole.

Table 3. Local modes of printed board

| Modal order of printed board | Frequency | Vibration mode position | Modal order of complete machine |
|-----------------------------|-----------|-------------------------|--------------------------------|
| 1                           | 324.83    | Z9                      | 1                             |
| 2                           | 473.19    | Z5                      | 3                             |
| 3                           | 474.36    | Z6                      | 4                             |
| 4                           | 521.78    | Z7                      | 7                             |
| 5                           | 604.21    | Z2/Z4                   | 9                             |
| 6                           | 630.48    | Z3                      | 13                            |
| 7                           | 662.37    | Z1                      | 14                            |
| 8                           | 751.15    | Z7/Z8                   | 21                            |
3.2. **Fundamental frequency check**

According to the octave range criterion [8], the fundamental frequency of the electronic product structure connected to the main structure of the star shall be greater than 100 Hz. The natural frequency of the chassis and the internal PCB module must be separated. At the same time, it shall meet the requirements of staggering the fundamental frequency of the printed board assembly and the whole machine. The fundamental frequency ratio shall meet the fundamental frequency ratio.
By analyzing the natural vibration characteristics of the whole machine structure and each printed board assembly structure, the corresponding natural frequency and vibration mode are obtained. The starting frequency of the whole structure is shown in figure 4, the fundamental frequency is 838.68 Hz, the fundamental frequency of the printed board is 324.83 Hz, and the fundamental frequency ratio is:

$$\frac{f_{\text{complete machine}}}{f_{\text{assembly}}} > 1.4 \quad \frac{f_{\text{complete machine}}}{f_{\text{assembly}}} < 0.7$$  \hspace{1cm} (1)$$

The fundamental frequency of the whole machine is relatively high compared with that of the whole satellite, which has good rigidity. The fundamental frequency of each printed board component and the whole machine are basically staggered to avoid resonance coupling.

4. Random vibration analysis

4.1. Simulation analysis

Based on the modal analysis, the modal superposition method is used to calculate the stress and displacement response of the structure under random vibration excitation in X, Y and Z directions, taking the damping coefficient of 0.03 and the test conditions shown in Table 2 as the load input. Figure 5 shows the structure in the three axes of X, Y and Z σ Stress and 3 σ Strain nephogram.

![Figure 5. Random vibration simulation results](image-url)
4.2. Result analysis and verification

4.2.1. Verification of printed board deformation
According to the reliability design requirements of spacecraft electronic products, the allowable limit of printed board deformation is:

\[ D_{\text{deformation}} = \frac{q_o}{b} \leq 0.006 \]  

(3)

Where: \( D_{\text{deformation}} \) is the deformation coefficient; \( q_o \) is the maximum deformation value of printed board, in mm; \( b \) is the length of the short side of the printed board, in mm.

It can be seen from Fig. 5 that the maximum deformation of printed board occurs on Z4 board in Y direction of random vibration analysis, with a size of 0.008mm. The check results are as follows:

\[ D_{\text{deformation}} = \frac{q_o}{b} = 0.00005 \leq 0.006 \]  

(4)

Meet safety requirements.

4.2.2. Structural strength check
The check formula of safety margin is:

\[ MS = \frac{[\sigma]}{\sigma_{\text{max}}} \cdot f - 1 \]  

(5)

Where: MS is safety margin, and MS of composite material is > 0.25; Chassis structure MS > 0; [\( \sigma \)] is the allowable stress; \( \sigma_{\text{max}} \) is the calculated maximum stress; F is the safety factor, taken as = 1.4. The check results are shown in Table 4:

| Name          | Allowable stress \([\sigma]/\text{Mpa}\) | Maximum stress \(\sigma_{\text{max}}/\text{Mpa}\) | Safety factor | MS  |
|---------------|----------------------------------------|---------------------------------|--------------|-----|
| Printed board | 338.4                                  | 13.4                            | 1.4          | 17  |
| Structural frame | 275                                   | 44.4                            | 1.4          | 3.4 |

It can be seen from table 4 that the structural safety margins of printed board and structural frame are 17 and 3.4 respectively, meeting the requirements of random vibration environment.

5. Sinusoidal vibration analysis

5.1. Sinusoidal vibration simulation results
Sinusoidal vibration analysis is mainly used to determine the steady-state response of the structure under the load varying sinusoidally with time. In sinusoidal vibration analysis, the excitation load is explicitly defined in the frequency domain, and the external load is known corresponding to each loading frequency. The external load can be force or forced motion (displacement, velocity or acceleration). Based on the results of modal analysis, this paper uses the harmonic response analysis method to simulate the sinusoidal vibration. In the input conditions given in Table 3, the external loads are acceleration and displacement respectively. During simulation calculation, harmonic response analysis shall be carried out in sections. The calculation results show that the response of the structure is the largest at 100 Hz, and the mechanical environment borne by the structure is the worst. Figure 6 shows the structural stress and strain nephogram of electronic equipment under the action of three axial sinusoidal vibration loads of X, Y and Z.
5.2. Result analysis and verification

5.2.1. Verification of printed board deformation

It can be seen from Figure 6 that the maximum deformation of the printed board occurs on the test unit Z1 board under the Z-direction sinusoidal vibration load, with a size of 0.039mm. The check results are as follows:

\[
D_{\text{deformation}} = \frac{q_o}{b} = 0.00027 \leq 0.006
\]  

Meet safety requirements.

5.2.2. Structural strength check

Under the action of X, Y and z-axis sinusoidal vibration load, the maximum stress of printed board and structural frame is 1.54 MPa and 4.45 MPa, which appears on the frame of test unit Z1 board and Z2 board under the action of Z-direction sinusoidal vibration load. The corresponding safety margins are 155 and 43 respectively, as shown in Table 5. The structural design meets the requirements of sinusoidal vibration environment.
Table 5. Safety margin results of sinusoidal vibration analysis

| Name          | Allowable stress $[\sigma]$/ Mpa | Maximum stress $\sigma_{\text{max}}$/ Mpa | Safety factor | MS |
|---------------|---------------------------------|-------------------------------------------|---------------|----|
| Printed board | 338.4                           | 1.54                                      | 1.4           | 155 |
| Structural frame | 275                            | 4.45                                      | 1.4           | 43  |

6. Conclusion
The structural stiffness and strength response of a spaceborne electronic equipment under different mechanical environments are studied. Firstly, the mechanical environment experienced by the structure is described; Then, the modal of the on-board electronic equipment is analyzed by finite element analysis, and its frequency and vibration mode are obtained; Finally, the mechanical analysis under sinusoidal vibration and random vibration is carried out, and the corresponding structural strength, stiffness response and safety margin data are obtained. The results show that the structural design of the electronic equipment meets the design requirements of spaceborne mechanical environment. The finite element analysis results are of great significance to the structural design of spaceborne electronic equipment, which can significantly reduce the development cost and shorten the development cycle.

References
[1] Zhang, R., Wang, K.W., Shen, Z.R. (2012) Application Research on reliability simulation test technology of high reliability electronic equipment. Electronic product reliability and environmental test, 3006: 13-19.
[2] Wu, M.W., Huang, C.J. (2015) Mechanical simulation analysis of a spaceborne electronic equipment. Electromechanical engineering, 3104: 49-56.
[3] Liu, C., Liu T.X., Jiang, W.J. (2016) Research on environmental design method of spacecraft electronic products against random vibration. Spacecraft engineering, 2503: 80-87
[4] Wu, W.Z., Cheng, L., Zhang, P. (2016) Finite element analysis of stiffness and strength of electronic equipment of a spaceborne radar. Electromechanical engineering, 3203: 56-59.
[5] Li, X.X., Yan, X.L. (2007) Modal analysis and anti vibration design of printed circuit board in electronic cabinet. Mechanical engineering and automation, 03: 19-22.
[6] Guan, Y.H, Xu, W.J. (2007) Dynamic simulation of a spaceborne electronic equipment. In: 2007 academic conference on mechanical electronics. KunMing. pp. 433-441.
[7] Fan, W.J. (2010) Broadband random vibration response analysis of spaceborne electronic equipment. Electronic and mechanical engineering, 26 (4): 5-8.
[8] Liu, Z.H., Yang, L., Dong, W. (2015) Application of octave range criterion in anti vibration structure design of an electronic equipment. Mechanical engineer, 02: 164-166.