IOP Conference Series: Materials Science and Engineering

Research on modeling method of honeycomb plate and its application on solar wing

To cite this article: Haitao Luo *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **563** 052020

IOP Conf. Series: Materials Science and Engineering **563** (2019) 052020 doi:10.1088/1757-899X/563/5/052020

Research on modeling method of honeycomb plate and its application on solar wing

Haitao Luo[1,2], Jia Fu[1,2], Haonan Wang[3], Yuxin Li[3]

[1] State Key Laboratory of Robotics, Shenyang Institute of Automation, Chinese Academy of Sciences, Shenyang, 110016, China;

[2] Institutes for Robotics and Intelligent Manufacturing, Chinese Academy of Sciences, Shenyang, 110016, China.

[3] Institute of Mechanical Engineering and Automation, Northeastern University, Shenyang 110819, Liaoning, China

* Correspondence: luohaitao@sia.cn

Abstract. Aerospace products need to simulate the complicated working conditions through dynamic simulation analysis to ensure the high reliability of product design. In this paper, the equivalent modeling method of honeycomb sandwich plate based on finite element analysis is studied, and the mechanical characteristic parameters of honeycomb core structure are obtained. At the same time, the composite components of satellite solar cell array are modeled and simulated. The modal analysis and frequency response analysis are carried out to provide an important basis for the design and improvement of the structure.

1. Introduction

Because of its large span, light weight, weak structural damping and low connection stiffness, the mechanical characteristics of solar cell array deployment system are very complex. We use dynamic response analysis to ensure high product stability. At present, composite materials are widely used in aerospace products. The finite element modeling of composites is very important to the accuracy of simulation results. The correct modeling method makes the results of the simulation analysis and the test results within the error range required to ensure that the mechanical properties of the product meet the technical requirements of the design [1].

2. Space Solar array finite element model

The solar cell array is composed of three battery panels connected with the bracket, and the battery arrays are connected by hinges. The main battery array is honeycomb sandwich plate, surrounded by carbon fiber frame.

Figure 1 shows the honeycomb sandwich plate structure. It is composed of the top and bottom two high-strength skin layer and the middle honeycomb sandwich layer, the surface layer is made of carbon fiber composite material, the sandwich structure is aluminum honeycomb structure [2-4].

IOP Conf. Series: Materials Science and Engineering **563** (2019) 052020 doi:10.1088/1757-899X/563/5/052020

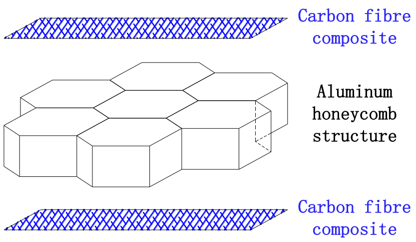


Figure 1. Schematic diagram of honeycomb sandwich plate structure

There are two parts in the simulation modeling of honeycomb sandwich structure. On the one hand, it is the finite element modeling of sandwich aluminum honeycomb structure, on the other hand, it is the finite element modeling of upper and lower carbon fiber composite panel.

The equivalent mechanical model of aluminum honeycomb should be considered because there is no cell library of honeycomb structure in MSC.Nastran space simulation and analysis software. For the hexagonal honeycomb plate, the sandwich plate theory is used to analyze the dynamics of the solar cell array. The sandwich layer of aluminum honeycomb is equivalent to an orthotropic layer with constant thickness. For hexagonal honeycomb [5-7], the equivalent elastic parameter is expressed as formula (1) - (5)：

$$E_x = E_y = \frac{4}{\sqrt{3}}\left(\frac{t}{l}\right)^3 E \tag{1}$$

$$G_{xy} = \frac{\sqrt{3}\gamma}{2}\left(\frac{t}{l}\right)^3 E \tag{2}$$

$$G_{xz} = \frac{\gamma}{\sqrt{3}}\frac{t}{l}G \tag{3}$$

$$G_{yz} = \frac{\sqrt{3}\gamma}{2}\frac{t}{l}G \tag{4}$$

$$\mu_{xy} = \frac{1}{3} \tag{5}$$

where, t is the thickness of the honeycomb cell panels, l is the length of the honeycomb cell panels, ρ is the density of aluminum honeycomb material, E is the elastic modulus of aluminum honeycomb material, G is the shear modulus of aluminum honeycomb material, μ is Poisson's ratio, γ is correction coefficient, generally take 0.4 ~ 0.6.

The upper and lower carbon fiber composite panel is defined by PCOMPG card. The schematic diagram of the carbon fiber panel is shown in figure 2. The area A of the board surface adopts 4 × 4 carbon fiber mesh layer with voids, and there are three mesh dense areas on the board surface, in which area B is encrypted by transverse carbon fiber, that is, 4 × 1 carbon fiber mesh layer with voids, and vertical carbon fiber is used in area C, which the void is also 4 × 1 carbon fiber mesh.

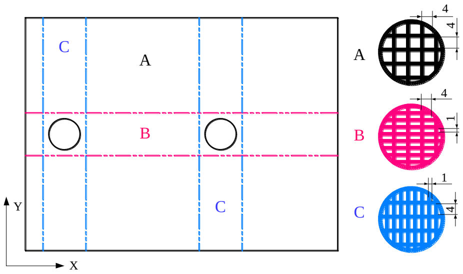


Figure 2. Schematic diagram of carbon fiber panel

The so-called carbon fiber mesh is laid out by a strand of fibers with a width of 2mm. For a 4 × 4 carbon fiber mesh layer with voids in area A, as shown in figure 3, the center distance of each fiber is 6 mm, which is staggered by 0 °and 90 °.In the finite element analysis software, it is not realistic to model the carbon fiber mesh according to the actual fishing net shape, and the isotropic layer of constant thickness is usually used to replace it [8-10].

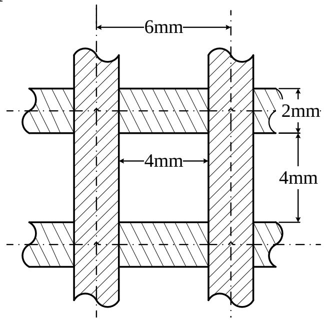


Figure 3. Schematic diagram of 4 × 4 carbon fiber grid

For the void 4 × 4 carbon fiber grid, if the thickness of a carbon fiber grid is 0.1 mm, it is equivalent to an anisotropic fiber layer of equal thickness, which is equivalent to flattening each strand of carbon fiber and filling the gap of 4mm with a strand of 2mm wide carbon fiber, In order to keep the mechanical properties of the panel unchanged, the total volume of each layer of carbon fiber is unchanged. Under this premise, the thickness of the anisotropic fiber layer is $1/3$ times that of the fiber thickness per share, that is, 0.03 mm.

Similarly, for the void 4 × 1 carbon fiber mesh, the width of the gap is 1 mm in the direction of encryption, which is equivalent to the width of the 2mm fiber to fill the gap of the 1mm. The equivalent thickness of the equivalent layer is $1/3$ that of the carbon fiber per share, that is, the thickness of the encryption direction is 0.06mm.

Table 1. Carbon fiber panel ply property sheet

Number of Plies	Actual overlay attribution		Simulation overlay attribution	
	Direction	Thickness	Direction	Thickness
1	0°	0.1mm	0°	0.06mm
2	90°	0.1mm	90°	0.03mm

The non-encryption direction, the gap width is still 4 mm, and the equivalent thickness of the layer is 0.03mm. Taking area B as an example, the X direction of the B area is defined as 0°, and the actual carbon fiber panel and the simulation analysis model are shown in table 1.

Because the solar wing installed on the star is two symmetrical structures relative to the star, the finite element simulation modeling of the solar wing is carried out with the single wing as the analysis target, and the compression points of the bracket and three solar cell arrays are taken as the fixed constraint boundary conditions. Figure 4 is a finite element model of a single-wing solar cell array [11-14].

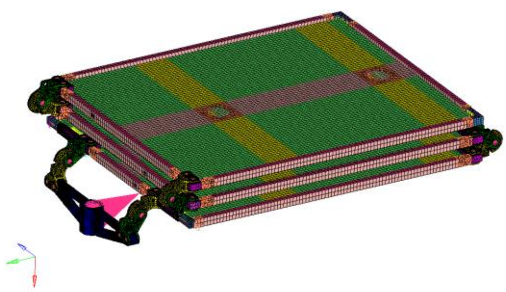


Figure 4. Finite element model of solar cell array

IOP Conf. Series: Materials Science and Engineering **563** (2019) 052020 doi:10.1088/1757-899X/563/5/052020

3. Space Solar array finite element model

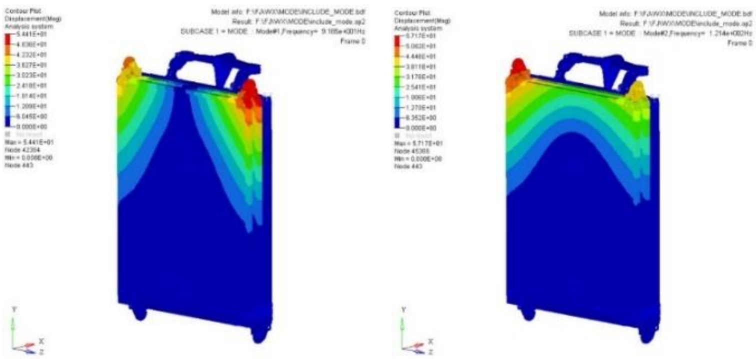
Once the natural frequency of machinery or structure is consistent with the frequency of external excitation resonance will occur which will lead to excessive deformation and even failure of machinery or structure. The objective of modal analysis is to identify the natural frequency of the system. In the field of aerospace, the fundamental frequency of solar wing collapse is strictly required by the design performance index [15-17].

MSC. Nastran provides three types of modal eigenvalue solutions, which are the tracking method, the transformation method and the Lanczos method, and the Lanczos method is the most effective for calculating a very large sparse matrix.

Through modal analysis, the first two order frequencies and modes of vibration are shown in table 2 when the solar array is closed. The first two mode shapes of the solar cell array are shown in figure 5.

Table 2. First two order frequencies and modes

Order	Frequency (Hz）	Mode of vibration
1	91.85	Reverse local vibration of outer hinges
2	121.4	Co-directional local vibration of outer hinges

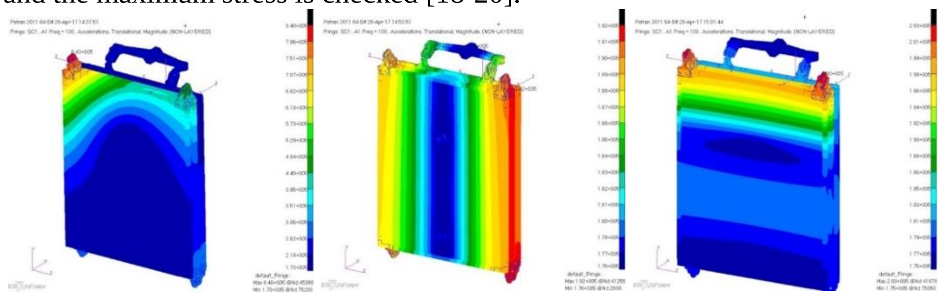


a) First-order mode cloud diagram b) Second-order mode cloud pattern

Figure 5. First two mode shapes cloud diagram

4. Frequency response analysis

In the rocket rising stage, the solar cell array is fixed on the turntable in the closed state. Under the dynamic load condition of the input acceleration varying with the frequency, the mechanical characteristics of the solar cell array in three directions are checked by modal frequency response method. Considering the specific working conditions, only the acceleration response within 100Hz is concerned, and the maximum stress is checked [18-20].



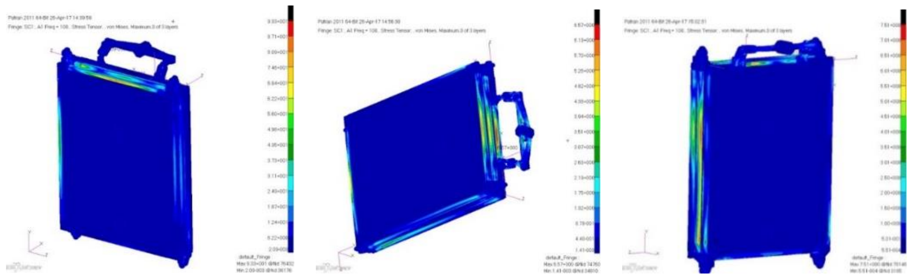
a) X maximum acceleration b) Y maximum acceleration c) Z maximum acceleration

Figure 6. Maximum acceleration response cloud chart for frequency response analysis

The cloud map of the maximum acceleration response in three directions is shown in figure 6. The maximum acceleration response size, frequency and response position are shown in table 3.

Table 3. Maximum acceleration response

Direction	Maximal acceleration (g)	Frequency (Hz)	Position
X	84	100	Solar cell out-of-array hinge
Y	19.2	100	Solar cell out-of-array hinge
Z	20.3	100	Solar cell out-of-array hinge



a) X maximum stress b) Y maximum stress c) Z direction maximum stress

Figure 7. Maximum stress response cloud diagram for frequency response analysis

Table 4. Maximum stress response

Direction	Maximum stress (MPa)	Frequency (Hz)	Position
X	93.3	100	Solar wing outer plate composite cross frame
Y	6.57	100	Solar wing inner plate composite cross frame
Z	7.51	100	Solar wing mid-plate composite vertical frame

The cloud diagram of the maximum stress response of solar array structures in three directions is shown in figure 7. The maximum stress, frequency and response position are shown in table 4.

The results of frequency response analysis show that the maximum stress of the frame composite is 93.3MPa, which is far less than the maximum stress limit of carbon fiber composites. The safety factor is large enough and the structure is safe.

5. Conclusion

In this paper, an accurate finite element model of solar cell array is established by modeling the sandwich aluminum honeycomb structure and carbon fiber composite panel in detail, taking a satellite solar cell array as the research object. Through modal analysis and frequency response analysis, the dynamic response characteristics of solar cell array single wing are checked. It provides a powerful modeling method support for the design and improvement of the whole model structure of solar array, and has important reference significance for the modeling of similar complex structures.

Acknowledgment

This research was financially supported by the National Natural Science Foundation of China (No.51505470) and Youth Innovation Promotion Association, CAS (2018237) and Jiang Xin-song Innovation Fund (20180504).

References

[1] L.Boldrin, S.Hummel, F.SCarpa etc 2016 Dynamic behavior of auxetic gradient composite hexagonal honeycombs, Composite Structure. 149, 114-124

[2] Shams A, Hegger J, Horstmann M 2014 An analytical model for sandwich panels made of textile-

reinforced concrete. Construction & Building Materials, 64(64) 451-459

[3] Cheon Y J, Kim H G 2015 An equivalent plate model for corrugated-core sandwich panels. Journal of Mechanical Science & Technology , 29(3) 1217-1223

[4] WANG Ningyang，SUN Hao，JIANG Hao 2016 On Grasp Strategy of Honeycomb PneuNets Soft Gripper, ROBOT, 38(3) 371-384

[5] Xianbing Li, Jihong Wen, Dianlong Yu 2012 The comparative study for mechanical equivalent method of honeycomb sandwich plate, Glass steel/composite materials. 1 11-15

[6] Baofeng Ji, Dongliang Chen, Weijie Sun etc, 2015 Yingwei Chen. “The equivalent analysis of the strength of the honeycomb sandwich plate based on the equivalent theory,” Computational materials science. 3(2) 12-15

[7] Xu A, Vodenitcharova T, Kabir K, et al. 2014 Finite element analysis of indentation of aluminium foam and sandwich panels with aluminium foam core. Materials Science & Engineering A, 599 125-133

[8] Ping L, Yan L, Xiong Z. 2015 Internal-structure-model based simulation research of shielding properties of honeycomb sandwich panel subjected to high-velocity impact. International Journal of Impact Engineering, 77 120-133

[9] Schaal C, Tai S, Mal A. 2017 On the assumption of transverse isotropy of a honeycomb sandwich panel for NDT applications. Society of Photo-optical Instrumentation Engineers.

[10] Colombo C, Carradó A, Palkowski H, et al. 2015 Impact behaviour of 3-layered metal-polymer-metal sandwich panels. Composite Structures, 133 140-147

[11] Sid-Ali K, Amar M, Boutaleb S, et al. 2014 Finite Element Prediction of Mechanical Behaviour under Bending of Honeycomb Sandwich Panels. Advanced Materials Research, 980(7) 81-85.

[12] Shiqiang Li, Xin Li, Zhihua Wang etc. 2016 Finite element analysis of sandwich panels with stepwise graded aluminum honeycomb cores under blast loading, Composites Part A. 80 1-12

[13] Ji W, Waas A M. 2015 Modeling Axial Impact Response of Sandwich Panels using Probability-Based Finite Element Analysis. Aiaa/asce/ahs/asc Structures, Structural Dynamics, & Materials Conference.

[14] Haitao Luo,Wei Wang,Jia Fu and Lichuang Jiao 2019 Finite Element Model Updating of Satellite Sailboard Based on Sensitivity Analysis, Shock and Vibration, DOI: 10.1155/2019/4547632

[15] T.Mukhopadhyay ， S.Adhikari. 2016 Free-Vibration Analysis of Sandwich Panels with Randomly Irregular Honeycomb Core, Journal of Engineering Mechanics. 142(11) 258

[16] Salem H, Boutchicha D, Boudjemai A. 2018 Modal analysis of the multi-shaped coupled honeycomb structures used in satellites structural design. International Journal on Interactive Design & Manufacturing, 12(3) 955-967

[17] Boudjemai A, Amri R, Mankour A, et al. 2012 Modal analysis and testing of hexagonal honeycomb plates used for satellite structural design. Materials & Design, 35 266-275

[18] Xin L, Wang Z, Feng Z, et al. 2014 Response of aluminium corrugated sandwich panels under air blast loadings: Experiment and numerical simulation. International Journal of Impact Engineering, 65 79-88

[19] Yahaya M A, Ruan D, Lu G, et al. 2015 Response of aluminium honeycomb sandwich panels subjected to foam projectile impact – An experimental study. International Journal of Impact Engineering, 75 100-109

[20] Li S, Xin L, Wang Z, et al. 2016 Finite element analysis of sandwich panels with stepwise graded aluminum honeycomb cores under blast loading. Composites Part A, 80 1-12