

Dynamic Simulation and Test Analysis of Space Truss Structure

H T Luo¹, P Wang², N Chen^{2,*}, S P Chen², W Wang², T K Wu³ and H N Wang²

¹Shenyang Institute of Automation Chinese Academy of Sciences, Liaoning, Shenyang 110016

²Northeastern University, Shenyang 110019, P.R. China

³Shenyang Ligong University, Shenyang 110159, P.R.China

Corresponding author and e-mail: N Chen, chenning2017@126.com

Abstract. Aerospace products are subjected to vibration environment during launching process. The dynamic characteristics of aerospace products play a decisive role in environmental adaptability of products. This paper describes the design of a space truss structure and the load, in order to get the dynamic characteristics, finite element modal analysis and modal test is done on the structural model, through correlation analysis to determine the degree of conformity of the finite element model with the experimental model, the truss structure and load finite element modal and experimental modal similarity degree is higher, the finite element the model is acceptable; The dynamic simulation of the structure is carried out by the qualified finite element model, and the dynamic simulation results are verified by the vibration test; The method for obtaining the dynamic characteristics of aerospace products and the way of dynamic simulation for launching process is of great reference value for the design of aerospace products.

1. Introduction

As the spacecraft moves towards a larger, more complicated direction. The space truss [1] is widely used because of its easy disassembly, good technology and light quality, and it can be adjusted according to specific needs. It is also an important part of the international space station. The application of space truss in space was first launched on August 20 and September 5, 1977 by Voyager 2 and 1 [2].

The random vibration is mainly caused by the exhaust noise of the engine during takeoff, the aerodynamic noise of the transonic flight section and the pressure pulsation in the engine combustion chamber. The low-frequency sinusoidal vibration [3] is primarily a free oscillation of the low order mode of the elastomers caused by pogo vibration and engine start, ignition quenching and the stage separation; the turbulence caused by the shock wave of the wind and transonic flight segments, and the low-order longitudinal oscillation caused by incomplete combustion of the engine.

This kind of low frequency vibration environment [4-5] can make the space truss structure damaged, loose connections, structural deformation, the phenomenon of performance degradation, and the vibration can reduce photoelectric instrument accuracy, mechanical fatigue, circuit instantaneous short circuit, open circuit, even function failure [6].

Therefore, in space product design, it is particularly important to obtain the accurate dynamic characteristics of the product, and the product dynamic characteristics are obtained mainly through the finite element simulation and test method, while the advantages of the finite element simulation are emerging.

If we can establish a benchmark finite element model that meets the requirements of engineering precision and can accurately reflect the dynamic characteristics of the space test device, it will undoubtedly play an important role in mastering the anti-vibration performance of the space products in the launch section, as well as the research on the effective load test conditions [7].

2. Design of truss and load structure

For space truss structure modeling and space load, as shown in Figure 1, Figure 2, the whole structure is made up of long tube, before and after the short tube, cover up and down, left and right sides cover plate, plate, connecting block many parts, fixtures and fittings, standard screw, weighs 15.848 kg, the material for AL7075.

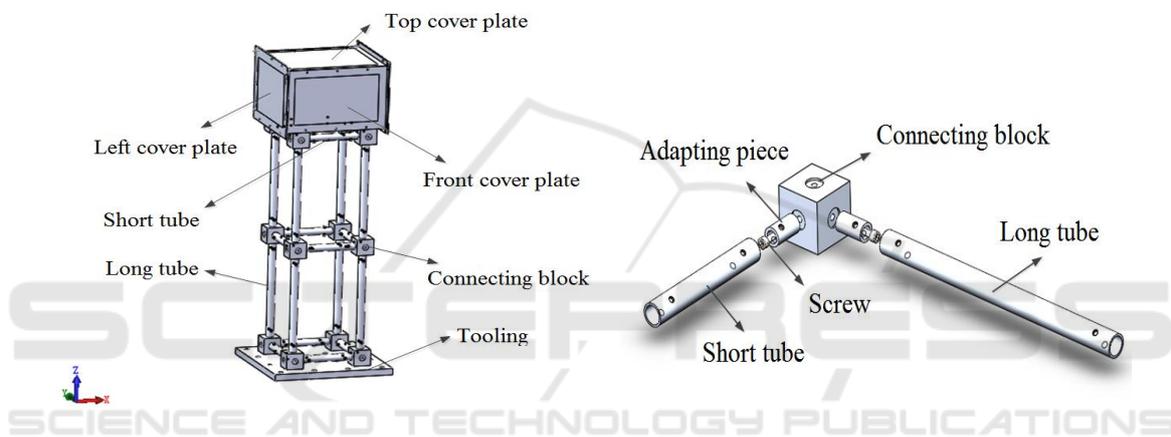


Figure 1. Three-dimensional model. **Figure 2.** The connection of connecting block.

The structure of the space load box is made of a plate-like structure, with a hollow structure inside, and a screw is used to connect the various surfaces. In addition to the eight faces of the structure of the box structure, the center of the other surfaces has a circular groove with a diameter of 18 mm and a depth of 2 mm. It is used for connecting pieces, connecting the long pipe, short pipe and connecting block through the connecting piece.

3. Dynamic characteristics of truss and load

3.1. Finite element modal analysis

Modal analysis is a prerequisite for kinetic analysis, because the natural frequency and the modality are the essential parameters of kinetic analysis, reflecting the vibrational properties of the structure. Modal analysis is divided into theoretical modal analysis and experimental modal analysis. In the finite element software, the theoretical modal analysis is carried out, which is also called computational modal analysis. The principle is to use the finite element method to discrete the structure, and then establish the vibration differential equation to solve the eigenvalue and eigenvector of the equation, namely the natural frequency and modal mode of the structure. The basic equation of modal analysis is:

$$[K]\{\Phi_i\} = \omega_i [M]\{\Phi_i\} \quad (1)$$

In this case, $[M]$ and $[K]$ are the mass matrix and the stiffness matrix, and ω_i and $\{\Phi_i\}$ are the natural frequencies and modalities of the first model of the structure. Modal analysis is to solve the equation (1) and obtain the natural frequency and modal mode of the structure.

The finite element model of truss and load adopts the right hand coordinate system, and the origin o is located at the center line of the lower cover plate. The X-axis is in the direction of the space load length, and the left cover plate points to the right cover plate. The Y-axis is in the direction of the space load width, and the front cover plate points to the rear cover plate, and the z-axis is straight up. The grid cells are triangular and quadrilateral units, and the connection squares and connections are divided by the body grid. As shown in Figure 3, the remaining parts are divided by the method of shell grid, the number of units is 338564, the number of nodes is 162717, and the model weighs 15.23 kg.

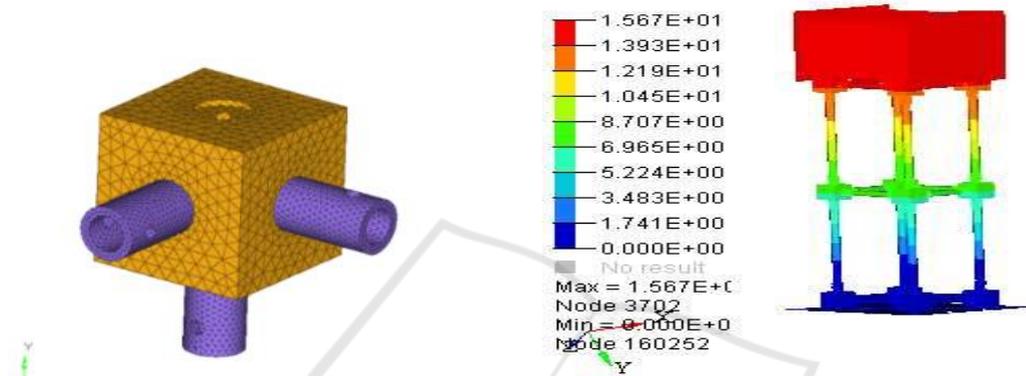


Figure 3. Connecting block and adapting piece. **Figure 4.** The first-order mode.

The finite element modal analysis of space truss and space load structure is carried out, and their natural frequencies are obtained, and the vibration mode of space truss and space load structure can be observed intuitively by the mode cloud graph. In this paper, we give the first four order modal type cloud image, as shown in Figure 4.

3.2. The modal test

The modal test of truss and load structure is carried out, and the truss structure is fixed to the vibration table. The constraint boundary of the simulated truss structure is shown in Figure 5. The test point layout is shown in Figure 6, with a total of 38 measurement points, and the excitation method is used to stimulate. The minimum elastic frequency of the clamping device used for support is much higher than the maximum analysis frequency of the specimen structure. Generally speaking, it is easy to realize the constraint boundary of small and medium structures, but it is difficult to realize the constraint boundary of large structures [8]. The quality of the truss structure in this paper is 15.85 kg, the analysis frequency is 200 Hz, the vibration table is 4500 kg, and the first-order resonant frequency is 2200 Hz, which satisfies the requirements of the truss structure constraint boundary.



Figure 5. Modal test.

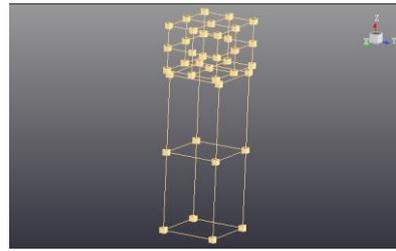


Figure 6. Geometric model.

The modal test analysis bandwidth of truss structure is 200 Hz, the frequency resolution is 1 Hz, and the steady-state diagram of the frequency response function is shown in Figure 7.

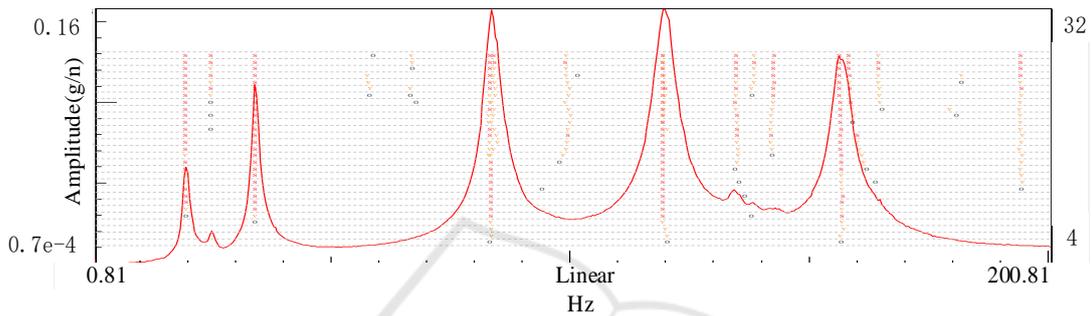


Figure 7. Steady state diagram of frequency response function.

3.3. Modal correlation analysis

In this paper, modal confidence criterion (MAC) in formula (2) is used as quantitative evaluation index.

$$MAC_{Test,FE} = \frac{[V_{Test}^T \cdot V_{FE}]^2}{[V_{Test}^T \cdot V_{test}] \cdot [V_{FE}^T \cdot V_{FE}]} \quad (2)$$

In the formula, V_{test} and V_{FE} are the modal vectors of test mode and finite element modal; the elements in $MAC_{Test,FE}$ matrix represent the cosine Angle between two modes, reflecting the relative degree of two modes. The MAC value should be between 0 and 1. When the MAC value is greater than 0.8, it can be considered that the vibration mode V_{test} of the test mode is similar to the mode V_{FE} of the finite element calculation mode; When the MAC value is less than 0.2, it can be considered that the vibration mode of the test mode and the mode of finite element calculation are orthogonal.

In this paper, LMS Virtual. Lab software is used to conduct correlation analysis based on the results of finite element modal analysis of truss and load structure and modal experimental results. As shown in Table 2, The finite element modal is similar to the experimental mode, and the first two order MAC values are all around 0.9, and the finite element model is acceptable.

4. Dynamic simulation and experiment

4.1. Dynamic simulation

The finite element model of the truss and the load structure has been proved to be acceptable. The finite element model is used to carry out the dynamic simulation analysis of the truss and the load structure X to the harmonic response dynamics, to simulate the low-frequency sinusoidal vibration of the rocket in the process of the rocket, and the conditions of the analysis are shown in Table 2, and the response measurement points are shown in Figure 8.

Table 1. Modal mac values and frequency differences of modes.

Order	Experimental modal	The simulation mode	MAC value	Frequency difference (Hz)
	Frequency(Hz)	frequency (Hz)		
1	19.7	25.5	0.819	-5.75
2	24.9	25.5	0.924	-0.51
3	34.1	32.3	0.669	1.83
4	83.6	99.9	0.519	-16.3

Table 2. Dynamic simulation condition.

Loading direction	Frequency range (Hz)	Amplitude 0-P
x dimension	4-10	5 mm
	10-200	1 g

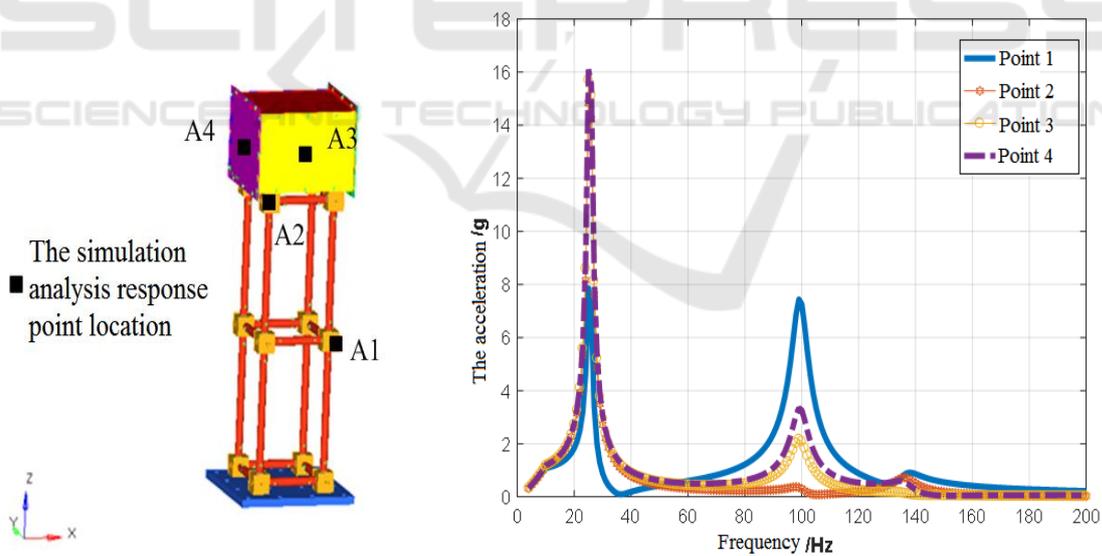


Figure 8. Response point position. **Figure 9.** Response point simulation results.

The simulation results of the four response points are shown in Figure 9, and the structure is most responsive at 25 Hz. The overall response of the structure at 25 Hz is shown in Figure 10 below. The maximum response of the structure is 16.6 g, which occurs at the top of the load.

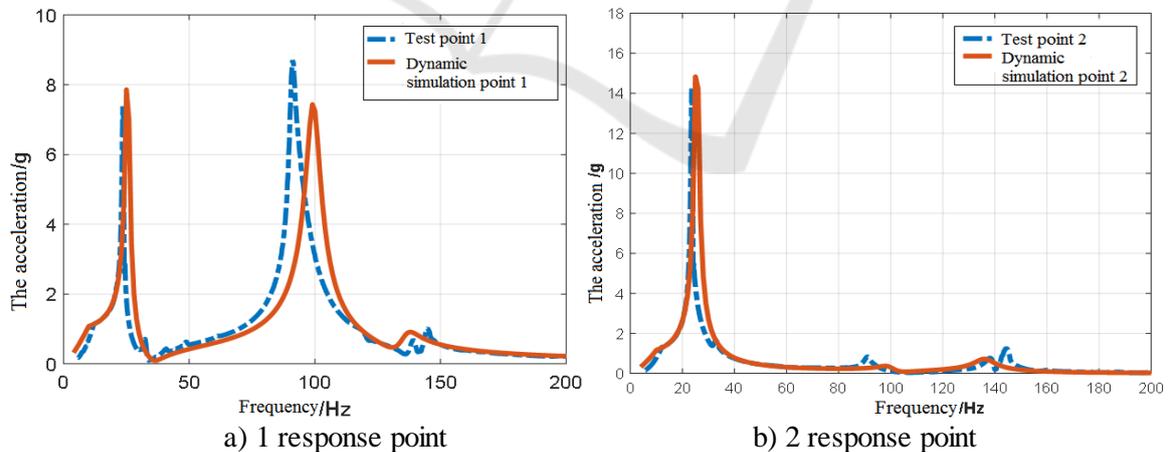
4.2. Dynamic test

The simulation results were verified by X - directional sinusoidal sweep vibration test of the space truss and load structure (as shown in Figure 10). The test conditions were consistent with the dynamic simulation conditions (shown in Table 2.), and the sweep rate was 4oct/min. The experiment uses two control points on the average control, and the position of the control points is located in the work and the truss structure, and the response point is consistent with the dynamics simulation response point.



Figure 10. Sinusoidal sweep vibration test.

The dynamic simulation results and the vibration test results of the four response points of the spatial truss and the load structure are shown in Figure 11. The frequency and amplitude response of the first order response of the four response points are shown in Table 3. The first order acceleration response frequency of the four test point frequency response analysis is larger than that of the vibration test, and the error is 5.4%. This is because the magnitude of the vibration test is larger, and it is difficult to avoid the phenomenon of loose and weak twisting between the components, which leads to the reduction of the stiffness. Through the comparison of the finite element simulation analysis and the vibration test data, it can be seen that the difference of the acceleration response value between the simulation and the test at the corresponding test point is small, and the finite element analysis data is a little bigger than the test data. From the data in Table 3, the first order error is 4.3%, the second order error is 2% , the third order error is 10% , and the fourth order error is 3.8% .



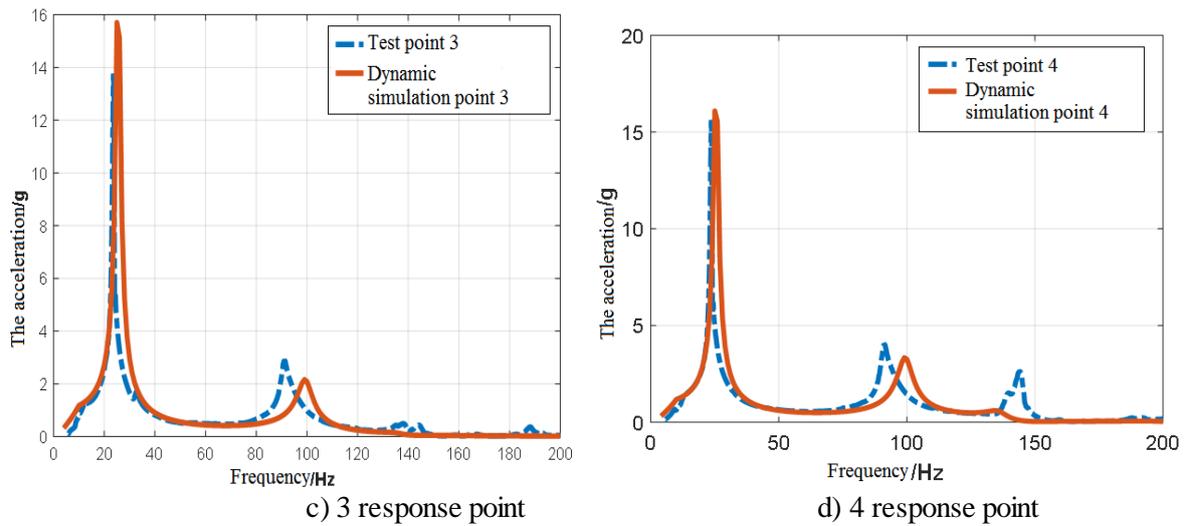


Figure 11. Comparison of dynamic simulation results and vibration test results.

Table 3. First order peak comparison of X direction and test.

Measurement point	A first order frequency			The first-order peak		
	The simulation (Hz)	Test (Hz)	Error (%)	The simulation (g)	Test (g)	Error (%)
1	25	23.7	5.4	7.86	7.53	4.3
2	25	23.7	5.4	14.83	14.53	2
3	25	23.7	5.4	15.8	14.3	10
4	25	23.7	5.4	16.1	15.6	3.8

5. Conclusions

In this paper, we design a space truss structure and load, in order to get the dynamics characteristics of the structure of the finite element modal analysis and modal test, through determining finite element modal and test modal similar degree is higher, the former two order MAC values are around 0.9, finite element model is acceptable; using the finite element model to perform the dynamic simulation of the structure, and using the vibration test to test the dynamics simulation results, the empirical evidence dynamics simulation results are less than the test results.

Acknowledgment

This work was supported by National Natural Science Foundation of China (No.51505470) and Youth Innovation Promotion Association, CAS.

References

[1] Zhang Y T 2014 Xi 'an China Design and simulation analysis of the deployable structure of fixed truss *Doctoral dissertation* Xi 'an university of electronic science and technology

[2] Li H F 2007 Changsha China Analysis of mechanical properties and passive vibration control of large flexible space truss *Doctoral dissertation* National University of Defense Technology

[3] Bao X D 1982 Analysis of low-frequency vibration environment conditions of foreign carrier rocket *Missile and space vehicle technology* Vol4 pp 0-6

[4] Wang Q Z and Liu B 1995 Research on prediction and control technology of space accident

- and dynamics environment *Environment technology* Vol 4 pp 1-6
- [5] Xia Y L and Wu J J 1998 Low-frequency vibration environment of space launch and its simulation *Intensity and environment* Vol 1 pp 1-8
- [6] Xu J H, Fan B and Lian H D 2015 The application of viscoelastic constrained damping in large aperture reflector *Space return and remote sensing* Vol 2 pp 32-38
- [7] Wang A P and Shang L B 2017 Analysis of modal analysis and model optimization of space science test device *Engineering and experiment* Vol 3
- [8] Li Z X, Li X Q and Chen X D 2008 Study on the influence of boundary conditions on modal influence in finite element analysis *Chinese mechanical engineering* Vol 19 pp 1083-1086

