Mechanical analysis and verification of the large-scale rack structure about the zero-gravity test of manipulator of the space station

Hailin Dai1, 2, Chengli Zhang1, Fanwei Meng1, Shaohua Meng1, Feng Gao1

1 Beijing Institute of Spacecraft Environment Engineering, No. 104 Friendship Road, Haidian District, Beijing, People’s Republic of China
E-mail: bhdhl88@163.com

Abstract: In this article, the mechanical properties of large-scale rack structures are analysed by MSC Marc analysis software. The test is carried out through modal testing. The modal frequency and modal shape are obtained through finite element analysis and LMS data acquisition system for modal test verification. It is shown that the modal shape of the modal test and the calculation by finite element analysis are consistent, which verifies the inherent characteristics of the frame structure effectively and provides reliability parameters of the manipulator for the space station in the zero-gravity test.

1 Introduction
The space manipulator in the third phase of the manned space station is a typical large flexible spatial mechanism. Zero-gravity tests on the ground should be carried out to validate the unlocking on-orbit performances of the space manipulator. A gravity suspension system with multi-degree of freedom should be developed to avoid product damage caused by the bending moment of gravity. As the support of the zero-gravity test system, the large-scale rack structure should be a proper structure to reach the high strength and the stability. Enough operation space is necessary as well, in order to ensure the structural stability of the whole test system and meet the test requirements for the natural frequency [1–4]. In this paper, an empty abdominal structure is designed to provide a support platform for the zero-gravity simulation test of the space manipulator on the ground.

The frame is the main load-carrying structure of the space manipulator. In order to avoid the resonance between the space manipulator and the manipulator products, the first-order modal of the frame structure should be >5 Hz, and the maximum equivalent stress should be <120 MPa under the rated load condition. Therefore, it is of great significance to study the mechanical properties, the modal frequency, and the modal shape of the large-scale rack structure.

2 Design of large-scale rack structure
The steel structure of the large-scale rack is designed according to the requirements of the zero-gravity test process on the ground. The requirements include lightweight, high stiffness, large operation space and suitable intrinsic property. Meanwhile, the main beam and stand bars of the frame structure are designed under the same rules. The configuration of the large-scale rack steel structure is shown in Fig. 1. The structure mainly includes the base, stand bars, main beam, bearing frame, deep beam, operating platform, and ladder. The main load-carrying parts are welded together by different specifications of steel plates in box-type structure and there are not any supports in the shell. Therefore, it can satisfy the load condition and the structure stiffness, at the meantime equipped with the operating platform and the security guard, which can facilitate the testing operation of space manipulator products at the top of the frame structure platform and ensure the safety of operators. An anti-slip gasket is installed under the base. The hard rubber is attached to the base and placed on the ground to increase the friction between the equipment and the ground, which can effectively prevent the ground from being damaged. The components are bolted to facilitate disassembly [5, 6].

In order to meet the requirement for the size of the internal enveloping space and the requirement for interference in the frame of the space manipulator during zero-gravity test, the overall dimension is 9348 × 8038 × 11,180 mm, as shown in Fig. 2.

3 Mechanical analysis of large-scale rack structure

3.1 Modelling of finite element analysis
The finite element model of the rack structure is established by MSC Marc analysis software. Due to the complexity of the frame structure, specific parts which do not affect the structural strength and the stiffness obviously are simplified, such as the screw hole, the pin hole, the chamfer and the stiffened plate and boss, in order to carry out the finite element mesh. When setting the material properties of the pretreatment, the unit selects kg, mm, and s. The large-scale rack structure studied in this paper belongs to the welded box-type structure; thus, the material should be carbon steel Q235-A, whose density is 7.860 kg/m³, modulus of elasticity is

![Fig. 1 Model of large-scale rack structure](image-url)
2.1 × 10^{11} \text{ Pa}, and Poisson's ratio is 0.3. The constraint is defined as
the contact between the frame and the foundation, and the
coefficient of friction is 0.1.

The frame structure is meshed. 3D shell unit is adopted in the
main body and free meshing is selected. The classification grade is
6 which is in high accuracy and satisfies the analysis requirements.
After intelligent meshing, the total number of nodes is 327,991 and
the total number of cells is 84,725 (Figs. 3 and 4).

3.2 Static analysis
During the zero-gravity test of the space manipulator, the load is
applied to the single-layer bearing frame and the double-layer
bearing on the top of the frame through the linear motion system.
The upper and the lower layers of the double-layer bearing frame
are, respectively, applied with 2000 N, and the single-layer bearing
frame is applied with 1000 N, which are all concentrated loads.
According to the actual load distribution of the rack structure, the
65,000 N load of the top platform frame is applied to the centroid
of the top frame, as well as the weight of the structure. The stress
simulation results show that the maximum equivalent stress is 20.1
MPa and the stress concentration occurs in the centre of the

---

Fig. 2 Size of large-scale rack structure

Fig. 3 Definition of geometric characteristics and meshing
double-layer frame, the end beam, and the sub-beam, as shown in Fig. 5 [7, 8].

The strain simulation results show that the maximum of the joint displacement (characteristic stiffness) is 1.2 mm, as shown in Fig. 6.

3.3 Modal analysis

The result of the above contact analysis is imported as an initial condition into the modal analysis to obtain the modal shape and modal frequencies of the rack when it is placed on the ground and loaded. The modal analysis results show that the first-order frequency of the support subsystem is 6.89 Hz, and the vibration is a sideways yaw. The second-order frequency of the system is the yaw in the direction of the door frame, and the frequency is 8.16 Hz. Figs. 7 and 8 show the modal shape after the deformation and the magnification.

4 Verification of modal test

In order to verify the correctness of the finite element analysis, results of LMS data acquisition system are adopted to carry out the modal test verification. The method of frequency domain identification of modal parameters based on weighted least squares and multiple-input and multiple-output (MIMO) transfer function, namely PolyMAX method, is used to extract modal frequencies and modal shapes. The system pole can be conveniently and clearly selected and identified on the stability map. Therefore, the modal characteristic parameters of the large-scale rack structure can be studied.

4.1 Experimental model

The modal shape of the frame structure is mainly obtained through the modal test. In order to completely and accurately obtain the various orders of the vibration structure under test and to satisfy the observability of the modality, the layout of the measuring points is particularly important. Meanwhile, it is also necessary to consider avoiding the position of the nodes. A total of 21 measuring points are arranged in the rack, among which are 4 in the base, 8 in stand...
4.2 Boundary conditions
In the modal test, the rack is placed on the foundation. A 5-mm polytetrafluoroethylene (PTFE) backing plate is laid between the stand bars and the foundation, and the coefficient of friction is about 0.1.

4.3 Excitation system
The force hammer excitation and the free suspension method are selected in the modal test. Combined with the finite element analysis model of the frame structure, an excitation point in each of the $X$, $Y$, $Z$ directions is selected. The position of the excitation point should be close to the node or pitch line and made uniform energy distribution to the structure.

4.4 Results
The frequency response function of each measuring point of the structure was obtained through the excitation system. The modal analysis of the structure was performed and the modal parameters of the frame structure were acquired. The modal shapes of the test results were consistent with the modal shapes calculated by finite element analysis. The modal test is as follows:

i. Horizontal $X$ direction: The first-order bending modal parameter is 6.229 Hz, and the damping is 1.56%; and the first-order torsional modal parameter is 9.12 Hz, and the damping is 0.27% (Fig. 10).

ii. Horizontal $Y$ direction: The first-order torsional modal parameter is 8.521 Hz, and the damping is 0.94%; and first-order translational modal parameter is 22.722 Hz, and the damping is 1.8% (Fig. 11).

5 Conclusion
In this paper, the mechanical characteristics of the large-scale rack structure of space manipulators are analysed and verified through zero-gravity tests. The results show that the modal frequencies and modal shapes obtained from the modal tests of the LMS data acquisition system are consistent with the modal shapes calculation from the finite element analysis. Large-scale rack structure design is reasonable, and the inherent characteristics of the structure reach the zero-gravity testing requirements on the ground for space manipulator products.

6 References
[1] Zheng, F., Chen, M.: 'New conceptual structure design for affordable space large deployable antenna', IEEE Trans. Antennas Propag., 2015, 4, pp. 1351–1358
[2] Wangu, L.: 'Tower testing stations in the world', Int. Electr. Power China, 2001, 1, p. 916
[3] Paul, Z.: 'Gossamer spacecraft membrane and inflatable structure technology for space applications' (Astronautics and Aeronautics, Reston, VA, USA, 2003)
[4] Cheng, W.M.: ‘Dynamic analysis of crane’, Hoist Convey. Mach., 2002, 2, pp. 24–26
[5] Changhua, H., Zheng, H.B.: ‘The development of the largest tower testing station in China’, Int. Electr. Power China, 2001, 1, pp. 14–15
[6] Osanm, M., Yuzhong, C., Yusuf, A.: ‘Expert spindle design system’, Int. J. Mach. Tool Manuf., 2005, 96, (45), pp. 2537–2548
[7] Santiago-Prowald, J., Ilia, H.: ‘Advances in deployable structures and surfaces for large apertures in space’, CEAS Space J., 2013, 5, (3–4), pp. 89–115
[8] Shihao, L., Wenhua, Y., Weifang, C., et al.: ‘Design plan optimum seeking of machine tool bed based on grey relational analysis’, J. Grey Syst., 2010, 22, (4), pp. 241–352

This is an open access article published by the IET under the Creative Commons Attribution License (http://creativecommons.org/licenses/by/3.0/)