Modal Analysis and Optimization of Some Internal Rotor Radar Stabilized Platform

Shan Xue[1], GuoHua Cao[2], and YuLong Song[3]

[1] Changchun university of science and technology, Changchun, China
xueshan@cust.edu.cn
[2] Changchun university of science and technology, Changchun, China
caoguohua@cust.edu.cn
[3] Changchun institute of optics, fine mechanics and physics,
Chinese academy of sciences, Changchun, China
lzy@cust.edu.cn

Abstract. The radar stabilized platform is the key part of radar supporting. It is used to isolate the loader movement and keep the radar relatively stable. The 3D model is built by Pro-E to aid the design of some internal rotor radar stabilized platform. The finite element model is built by Ansys after the 3D model is simplified. The modal of platform is analyzed through finite element theory and Ansys software. And the natural frequency and vibration catalog of the eighth modal for stabilized platform is acquired. The modal of different mesh and different materials are compared. It provides references for like analysis. And it also provides proof for optimization of stabilized platform. The program is designed to improve dynamic performance of stabilized platform.

Keywords: Internal rotor, Stabilized platform, Modal analysis.

1 Introduction

The radar is difficult to be stabilized due to the naval swing interference. Emergence of stabilized platform could keep the radar stationary isolating naval motion. Study on the structure, material and transfer method of stabilized platform is becoming more and more important. The development trend is light weight, small volume, high accuracy, large rigidity, and good dynamic performance and manufacture ability.

As the modern method to study the dynamic character of structure, the modal analysis is the most basic and important part of structure design, which provides one powerful tool to design and evaluate structure of product[1].

The body model of some internal rotor radar stabilized platform is built in Pro-E4.0 it is input Ansys12.1 after simplification. The modal is analyzed through finite theory and Ansys software. The eighth modal of nature frequency and vibration catalog is acquired. And the different modals and weights from different meshes and materials are compared. It provides reliable proof for the optimization of platform.

D. Jin and S. Lin (Eds.): CSISE 2011, AISC 104, pp. 517–523, 2011.
springerlink.com © Springer-Verlag Berlin Heidelberg 2011

2 Digital Model Building of Some Internal Rotor Radar Stabilized Platform

The designed some internal rotor radar stabilized platform is composed of radar load stage, stator of pitch axis internal rotor moment electric engine, rotor of pitch axis internal rotor moment electric engine, pitch axis shell, azimuth axis shell, rotor of azimuth axis internal rotor moment electric engine, stator of azimuth axis internal rotor moment electric engine, base, shaft bearing, encoder, load stage, radar antenna, screw fastener and optical fiber gyro[2].

The digital model for structure of external rotor stabilized platform is built by PRO-E4.0. The model is shown as Fig.1.

3 The Finite Element Model Building of Some Internal Rotor Radar Stabilized Platform

The body model of some internal rotor radar stabilized platform in Pro-E4.0 is input Ansys12.1 after it is simplified. The shell and body are both applied in the finite model.

3.1 Simplification of Model

The components number of stabilized platform is large and the connecting relationship is complicated. So when the stabilized platform modal is analyzed, some components in structure should be simplified appropriately to improve computing speed with precondition of meeting computing accuracy. The simplification of structure is the key factor of finite model building.

A. Simplification of screw connecting

According to system character, the screw connecting rigidity is judged from modal analysis of components. The flexible or rigid connection is chosen. Through analysis, there is good rigidity in dense screw connection.

The rigid component is rigid connected in ansys operate environment. If there is less screw connection between two components, the connection will own bigger flexible effect. The connection parts are equivalent as dynamic model with many springs. The equivalent spring rigidity could be calculated by integration[1].

B. Simplification of bearing

Simplification of bearing is mainly for the bearing ball. The radial bearing ball could be simulated by 3D spherical body, 3D spatial bar and 3D spaces according practical conditions in engineering analysis. The bearing ball is simulated by 3D spatial bar in finite element model building. The internal and external parts of bearing are classified with body element. The internal and external parts are cemented. The external cage is cemented with the frame. And the longitudinal location of bearing is assured[3].

C. Other simplification

The aperture, beveling and rounding smaller than 2mm are omitted. The shaft is simulated by girder element. The bar is simulated by bar element.

3.2 Mesh Division of Model

The mesh division is very key in finite element analysis. According to the structure of stabilized platform, the meshs are divided in Ansys. The division method of all tetrahedron and combination of scanner and tetrahedron are both applied as shown in Fig.2 and Fig.3.

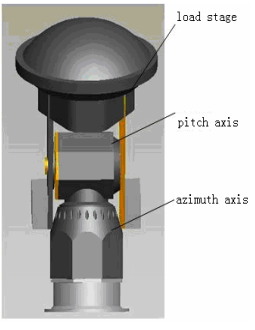

Fig. 1. Sketch of internal rotor stabilized platform structure

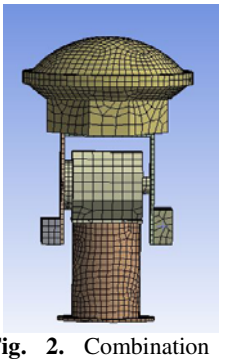

Fig. 2. Combination of scanner and tetrahedron method

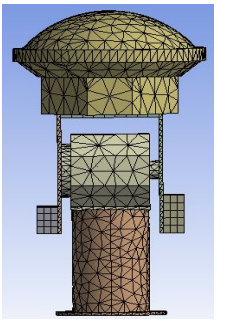

Fig. 3. All tetrahedron method

The division speed of all tetrahedron is fast with less error. But the division quality of meshs is not good. The demand for structure is higher in hexahedron division with good quality of meshs.

The meshs distortion of two methods is 0.45 and 0.86 respectively. The division method of combination of scanner and tetrahedron is preferred than all tetrahedron method.

3.3 Material of Stabilized Platform

The modal of stabilized platform with different material is analyzed.

It is for design and optimization of stabilized platform. The stabilized platform is designed with all parts steel. And it is also designed with part duralumin and combination of bearing and steel. The structure material is shown as Table 1.

Table 1. Parameters of structure material

No.	Material	E/MP	μ	ρ(10^{-3}g/mm^3)
1	2A12	72000	0.33	2.80
2	GCr15	207000	0.29	7.81
3	Q236	196000	0.3	7.85

4 Modal Analysis

The modal analysis is the most basic and important part of analysis of dynamic character. It is the modern method to study the dynamic character of structure. And it is the powerful tool to design and evaluate structure of product. The natural frequency and deformation amplitude of system could be acquired by modal analysis. The rigidity could be analyzed and the resonance vibration could be avoided[4].

From the variation principle of elastic mechanics, the dynamic balance equation of stabilized platform is as following.

$$[M]\{\ddot{u}\}+[C]\{\dot{u}\}+[K]\{u\}=\{P(t)\}+\{N\}+\{Q\} \tag{1}$$

In the expression, $[M]$ is the mass matrix, $[C]$ is the amortization matrix, $[K]$ is rigidity matrix, $\{P(t)\}$ is the external force function vector, $\{N\}$ is the nonlinear external force vector related with $\{\dot{u}\}$ and $\{u\}$, $\{Q\}$ is the boundary constrain counterforce vector, $\{u\}$ is the shifting vector, $\{\dot{u}\}$ is the speed vector, $\{\ddot{u}\}$ is the acceleration vector.

To solve the natural frequency and vibration catalog of stabilized platform, the external force and amortization is zero. The right side of the equation is equal to zero,

$$[M]\{\ddot{u}\}+[K]\{u\}=\{0\} \tag{2}$$

And the corresponding characteristic matrix equation is,

$$\left([K]-\omega^2[M]\right)\{u\}=\{0\} \tag{3}$$

In the expression, ω is the natural frequency.

Because the amplitude of free vibration is not zero, i.e. $\{u\}\neq 0$, the deformation of Eq.(3) will be as Eq.(4).

$$[K]-\omega^2[M]=0 \tag{4}$$

Then ω_1^2, $\omega_2^2 \ldots \omega_n^2$ and $\{\Phi_1\}$, $\{\Phi_2\}\ldots\{\Phi_n\}$ could be solved. ω_i and Φ_i is the natural frequency and vibration catalog of the ith modal. The natural frequency and vibration catalog represent for the dynamic characters.

The above equations are solved by Ansys[5]. The modal of stabilized platform with different material is analyzed with Ansys. The comparison of the natural frequency of the 8th modal is shown as Table2. The first and second natural modal is shown I Fig.4, Fig.5, Fig.6 and Fig.7. Stabilized platform with part duralumin and bearing steel is analyzed. The natural frequency and vibration catalog of modal is described as Table.3. The weight of stabilized platform with part duralumin and bearing steel is 153kg. but the weight of Stabilized platform with all steel is 193kg.

Table 2. 8th natural frequency of modal comparison for stabilized platform with different material

	Stabilized platform with part duraluminand bering steel（HZ）	Stabilized platform with all steel（HZ）
1st natural frequency	98.22	88.56
2nd natural frequency	145.37	95.56
3rd natural frequency	150.73	102.56
4th natural frequency	260.57	245.97
5th natural frequency	314.18	299.90
6th natural frequency	372.88	314.23
7th natural frequency	389.39	357.99
8th natural frequency	469.68	484.31

Table 3. Modal of stabilized platform with Stabilized platform with part duralumin and bering steel

Order of modal	natural frequency（HZ）	Vibration description
1	98.22	Rotate around pitch axis
2	145.37	Partial swing with right balance weight
3	150.73	Swing around azimuth axis
4	260.57	Rotate around azimuth axis
5	314.18	Rotate around pitch axis +partial bending
6	372.88	Partial swing with right balance weight
7	389.39	Bending perpendicular to pitch axis plane
8	469.68	Up-and-down movement

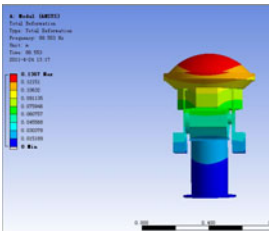

Fig. 4. The 1st modal of Stabilized platform with all steel

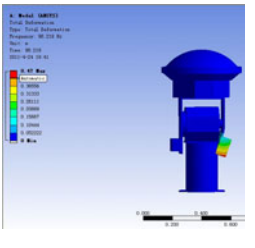

Fig. 5. The 1st modal of stabilized platform with part duralumin and bearing steel

The weight of stabilized platform with part duralumin and bearing steel is lightened by 20.7%. And the natural frequency is enlarged from 88.56 to 98.22. The rigidity is also intensified with the natural frequency enlargement.

It is shown in the modal analysis that in some internal rotor stabilized platform, material of the minor part could be duralumin, and the precise part should be bearing steel. The weight could be lightened and the rigidity could be intensified.

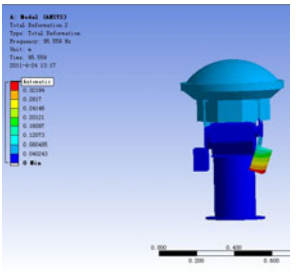

Fig. 6. The 2nd modal of Stabilized platform with all steel

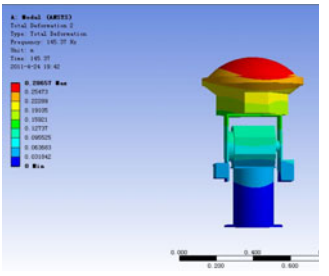

Fig. 7. The 2nd modal of stabilized platform with part duralumin and bering steel

5 Summary

(1)A kind of some internal rotor stabilized platform is designed and the 3D body is built by Pro-E.

(2) The body model is input Ansys software for finite element model building after simplification. The connections are simplified correspondingly.

(3)The modal is analyzed with finite element theory and Ansys software. The natural frequency and vibration catalog of 8th modal is acquired. Modal and weight with different meshs and material are compared.

(4) The division method of combination of scanner and tetrahedron is preferred than all tetrahedron method. Material of the minor part could be duralumin, and the precise part should be bearing steel. The weight could be lightened and the rigidity could be intensified.

The design of radar stabilized platform is optimized through modal analysis. And the dynamic character is improved in large amount. It provides good references for design of like device.

References

1. Duan, Y., Gu, J., Ping, L.: Modal analysis and experimental study on radar antenna stage. Mechanical Design and Fabrication 2(2), 214–216 (2010)
2. Russell, J.C.: A Control System and Application Program Interface (API) for an Elevation Over Azimuth Tracking Pedestal. A dissertation submitted to the Department of Electrical Engineering,University of Cape Town, in partial fulfilment of the requirements for the degree of Bachelor of Science in Electrical and Computer Engineering (October 20, 2008)

3. Yang, X., Jia, H., Feng, C.: Finite Element Modal Analysis of the Gyroscope-Stab ilized Platform. Computer simulation 25(9), 70–73 (2008)
4. Chen, G., Han, X., Liu, G.: Dynamic character analysis and optimizaion of some heavy duty commecial vehicle operator cabin. Chinese Mechanical Engineering 10(20), 2509–2513 (2010)
5. Zhang, c.: Engineering application examples of Ansys structure ananlysis. Mechanical industry press (2010)