Structural Engineering Analysis for a Control Moment Gyroscope Framework

TIAN Xiu¹², LANG Yue-dong¹²*, LAI Xin-xiao², HOU Er-yong³

¹ Eastern Theater General Hospital, Nanjing, China
² Beijing Aerospace Control Instrument Research Institute, Beijing, China
³ National University of Defense Technology, Changsha, China

Corresponding author: langyuedong9910@163.com*

Abstract. The framework structure of a control moment gyroscope (CMG) system was regarded as a research object, which stiffness and vibration mode was analyzed by ANSYS finite element method software, so that its static and modal characteristics under a large load were obtained. And according to the analysis results, its structural optimization design was implemented to meet the requirements of structural characteristics in the frame system. The results of optimized analysis showed that the structure strength and dynamic characteristics were enhanced obviously, which verified the necessity and reasonability of engineering analysis for the structural design. Therefore, it can provide a theoretical basis for system design.

1. Introduction

The control moment gyroscope (CMG) is the key actuator for the attitude control of long-running spacecraft such as space station[1]. It has the characteristics of large output Torque, fast dynamic response, good control linearity and high efficiency, so it is widely used in aerospace field.

The CMG is composed of momentum flywheel with high rotating speed, gyroscope room supporting the flywheel, high-speed drive motor, frame (including servo drive system) and so on. Its frame is used to support the gyro house and drive the gyro house to rotate. The angular momentum of momentum flywheel is forced to change direction by the rotation of the gyro house, so that the larger control torque is output and the attitude adjustment of the spacecraft is realized. A schematic diagram of how it works is shown in Fig.1. Among them, the Gyro $G_z$ is used to sense the angular velocity $\omega_z$ of the gyro house, and its signal is converted, amplified and corrected to feed back to the torque motor $M_z$ on the vertical axis, so that the motor outputs the control torque and drives the gyro house to rotate, then the rotating transformer $T_z$ on the vertical axis can detect the relative rotation angle and realize the closed-loop control of the system. Because of the limitation of the load-bearing capacity and the installation space of the spacecraft, the CMG is required to have a light weight, a compact structure and a higher load capacity, it should also have high stiffness and resonant frequency to meet the bandwidth requirements of the system. Therefore, it is necessary to analyze and verify its stiffness and modal characteristics in the design process. In this paper, the static and modal analysis of the gimbal structure of CMG is carried out by using the finite element method, which provides a theoretical basis for the optimal design of the gimbal structure.
2. Basic Structure of Control Moment Gyro Gimbal

The CMG gimbal system is uniaxial driven and its three-dimensional structure is shown in Fig. 2. The basic structure of the system includes 5 parts: frame, motor component, rotary component and control box. The frame is designed into two parts, the upper bracket and the lower base, so that the gyro room can be installed on the frame base.

As the basic external support of CMG system, the frame is the largest part in the whole system in volume and mass. Its own weight does not exceed 90 kg, but it needs to bear the weight of the gyro room equipment up to 160 kg, its load-to-weight ratio is about 2:1. Therefore, it is necessary to carry out the statics analysis of the frame structure in order to ensure its high rigidity and strength[2,3]. In addition, the resonant frequency of the frame is also a problem to be considered in the structural design, in order to avoid resonance at certain resonant frequencies[4～6]. Therefore, in the design process, as far as possible to increase the resonant frequency of the frame to a certain range, thus improving the reliability of the whole system.

3. Structural Analysis

In this paper, the static and Modal Analysis of CMG frame is carried out by ANSYS software. In order to simplify the modeling process, the solid model was built under Pro/e software, and was directly imported into ANSYS through the conversion interface, thus a complete analysis model was obtained.

In the process of model analysis, in order to reduce the difficulty of Mesh generation and calculation, the characteristics of screw hole and fillet are removed from the model, and the upper and lower parts of the outer frame are simplified as a whole for mechanical analysis. The material properties of the model are linear, elastic, isotropic and do not change with temperature. The material of the outer frame is cast aluminum ZL201, and its material properties are shown in Table 1.

| Name of material      | Density(Kg/m³) | Poisson   | Young's modulus(Pa) |
|-----------------------|----------------|-----------|----------------------|
| CAST aluminum ZL201   | 2700           | 0.33      | 7.1×10¹⁰             |

The unit type is Solid45, and the grid unit is 8-node hexahedron structure, which is often used in entity analysis. The advantages of SOLID45 are regular grid division, few nodes, few errors and moderate computation. For the application of load and restraint conditions, since both the upper and lower bases of the frame are assembled with shaft end assemblies, the force on the frame is mainly the weight of 160Kg of the gyro house carried by these two parts, thus a load is applied on two end faces of the upper bracket and the lower base. The frame is connected and fixed with the base through the bottom, and a constraint is imposed on the lower surface of the base. A preliminary preprocessing model for the framework is shown in Fig.3.
4. Calculation Results and Analysis
The finite element software ANSYS is used to carry out the mechanical analysis of the frame, and the static and vibration modal analysis results of the frame are obtained.

4.1. Statics Analysis and Optimization
The statics simulation results are shown in figure 4. When the upper support and the lower base of the frame system are relatively fixed, a load of 160kg is applied, and the stress distribution of the frame under 3g acceleration in the direction of gravity is shown in Fig. 4.

(a) Stress distribution  (b) Strain distribution

Fig. 4 Results of statics analysis of frames

(a) the maximum compressive stress is 11.7 MPa and the maximum tensile stress is 2.18 MPa. According to the allowable stress 295MPa of cast aluminum ZL201, the Factor of safety of the frame structure is about 25. The strain distribution of the frame is shown in Fig. 4(b) . The maximum deformation occurs at the root of the frame. The maximum compressive strain is 0.078 mm, and the maximum tensile strain is 0.03mm. On this basis, the structure of the frame is optimized, and stiffeners are added to the weak part of the base of the frame to improve its stiffness.

The stress distribution of the frame is shown in Fig. 5(a) . Under the same loads and constraints, the maximum compressive stress and tensile stress of the optimized structure are 2.6 MPa and 0.92 MPa respectively. The Factor of safety of the frame structure is increased to 113. The strain distribution of the optimized frame is shown in Fig. 5(b) . The maximum compressive strain is 0.018mm, and the maximum tensile strain is 0.008mm.
4.2. Vibration Modal Analysis and Optimization

On the basis of the static analysis of the frame, the vibration mode of the frame is analyzed, and the resonant frequency in the frequency range from 0 Hz to 2000 Hz is analyzed. As can be seen from Fig. 6(a), the first-order resonance frequency obtained is 69.8 Hz and lower. As can be seen from Fig. 6(B), the first order resonance frequency of the optimized structure is 104.95 Hz, and the improved resonance frequency has been greatly increased.

| Mode order | Before optimization | After optimization |
|-----------|---------------------|--------------------|
| First     | 69.76               | 104.95             |
| Second    | 135.16              | 187.76             |
| Third     | 153.83              | 221.6              |
| Fourth    | 615.04              | 627.36             |
| Fifth     | 638.23              | 680.7              |

The results of the 5-order Modal analysis before and after the frame structure optimization are shown in Table 2.
From the data in Table 2, it can be seen that the resonance frequency of the optimized frame has been increased, which indicates that the dynamic characteristics of the optimized frame have been greatly improved. After mechanical analysis and optimum design of the frame structure, a prototype of the frame system is developed. Its mass is about 90 kg. Under the condition of bearing 160 kg load, the system runs smoothly with high control accuracy and the position accuracy 20" can be achieved. The vibration and noise of the system are controlled in the effective range, and the reliability is high, which can meet the requirements of the system.

5. Conclusion
CMG is one of the key actuators for attitude control of large spacecraft. In this paper, the static stiffness and vibration modes of the CMG are analyzed by ANSYS, and the weak links are optimized. The results of optimization analysis show that the stiffness and strength of the improved frame structure are greatly improved, and the dynamic characteristics are also improved, which effectively verifies the validity and rationality of the analysis results. Therefore, it is necessary to analyze the engineering mechanics of the CMG. The analysis method in this paper can provide some theoretical basis and reference value for the design of CMG.

References
[1] Feng Hongwei, the house was built. Modal Analysis and experimental research on high-speed Rotor of maglev control moment gyroscope. Chinese Journal of Inertial Technology. 2005,2(13) : 72 ~ 75.
[2] Liu Zhenghua, Er Lianjie, Xu Chunmei. Finite Element analysis of three-axis flight simulator. Mechanical design and manufacture, 2002,10(5) : 7 ~ 9.
[3] Liang Yingchun. Finite element optimization design of frame structure of three-axis simulator. Chinese Journal of Inertial Technology, 2000.6(4) : 81 ~ 83.
[4] Yang Pengjun, Wang Jiamin. Finite Element analysis of inertial platform components [ J ] . Mechanical and electronic, 2004(4) : 15-18.
[5] Gan Zhihong and Zhang Bao. Engineering analysis of frame structure of airborne electro-optical stabilized platform [ J ]. Optical precision engineering, 2008,16(12) : 2442 ~ 2446.
[6] Fang Jiancheng, Qi Zihui, Zhong Mai Ying. Feedforward compensation method for unbalanced moment of three-axis inertial stabilization platform for aerial remote sensing [J]. Chinese Journal of inertial technology,2010,18(1):38~43.
[7] Hilkert J M. Inertially stabilized platform technology: Concepts and principles[J]. IEEE Control Systems Magazine, 2008(12): 26~46.