Stiffness and Mode Analysis of Framework for an Aerial Inertial Stabilized Platform

Yuedong Lang¹,²,*, Xiu Tian¹,²,a, Yang Kai³,b, Eryong Hou⁴,c
¹ Eastern Theater General Hospital, Nanjing, China
² Beijing Institute of Aerospace Control Instruments, Beijing, China
³ Department of Aeronautical mechanical engineering Army Aviation Academy, Beijing, China
⁴ National University of Defense Technology, ChangSha, China

*Corresponding author: langyuedong9910@163.com
*a tianxiu999@163.com
*b yangkai4545@163.com
*c houeryong1@163.com

Abstract—In the framework structure design of an aerial inertial stabilized platform (AISP), simulation software was used to analyze its stiffness and vibration mode. Its stress and strain distributions and modal characteristics under a large load were obtained. Then, according to the analysis results, its structure optimization design was implemented to meet the requirements of AISP in control precision. The results of optimized analysis showed that the rigidity of optimized frame structure was significantly improved, while its dynamic characteristics were also effectively improved. Therefore, it verified the necessity and feasibility of engineering analysis for the platform structure design, and it could provide a theoretical basis for AISP design.

1. Introduction
The AISP is a kind of attitude stabilization equipment with the functions of air target acquisition, tracking and remote sensing surveying and mapping[1]. It can effectively isolate the motion interference of the carrier, improve the stability accuracy of the optical system, so as to ensure the high definition of the captured image. The research of inertial stabilization platform is earlier in many countries, and a lot of work has been done in modeling and analysis[2-3]. At present, there are mature platform products, such as pav30/80 of Leica, T-as and gsm3000. With the continuous improvement of the level of aviation industry in our country, the demand for high-precision, large load-carrying capacity inertial stability platform is more urgent[4-6].

Due to the limitation of the aircraft's carrying capacity, the frame structure of the inertial stabilization platform is required to have both light weight and high load capacity, and its load usually needs to reach more than 2 times of its own weight. At the same time, it must also have high stiffness and resonant frequency, so that the platform has to provide enough bandwidth to meet the control accuracy requirements of the stable platform. Therefore, it is very necessary to carry out engineering analysis and optimization of the frame structure in the design of the stable platform. In this paper, the
finite element method is used to analyze the stiffness and modal characteristics of the platform frame structure, and its optimal design is carried out, so as to meet the practical application requirements.

2. The basic principle and structural design of the platform

2.1. Schematic diagram of platform structure layout

Aviation inertial stabilization platforms usually adopt a two-degree-of-freedom frame structure. The platform includes two closed loops of pitch and roll. Its working principle is shown in Figure 1.

Among them, gyro $G_i(i=x,y)$ is used to sense the angular velocity of the two rings $w_i(i=x,y)$. Its signal is fed back to the torque motor $M_i(i=f,r)$ on the corresponding shaft after conversion, amplification and correction. The motor outputs control torque to drive the ring frame to rotate, then the rotary transformer $T_i(i=f,r)$ on the corresponding shaft can detect the relative rotation angle.

The stable platform mainly consists of inner ring components (including sensors, gyroscopes, acceleration sensor, etc.), platform frame (including drive motor, circuit board, transmission gear, etc.), support frame (including four shock absorbers), base plate and shaft end components, etc. as shown in Figure 2, optical load is usually installed on the inner ring components.

![Figure 1. schematic diagram of working principle of inertial stabilization platform layout](image1)

2.2. The structure of the platform

As the basic external support of the inertial stabilization platform, the frame is the part with the largest volume and mass in the whole system. It can bear the optical load and the total weight of the shaft end assembly up to 160kg, but its own weight is not more than 40kg, and its ratio of load to dead weight is as high as 4:1. Therefore, in the design of stable platform, it is necessary to carry out engineering mechanical analysis on its frame to make it have sufficient rigidity to meet the load requirements. On this basis, it is necessary to improve the stiffness and resonant frequency of the frame as much as possible, so as to improve the control accuracy and reliability of the stable platform.

![Figure 2. three-dimensional structure diagram of inertial stabilization platform](image2)
3. Establishment of Calculation Model

Firstly, a calculation model is established to carry out static and modal analysis on the frame of the inertial stabilization platform. The framework model in the simulation software is directly imported through the conversion interface. In order to reduce the difficulty of grid division and computation, the features of threaded holes and fillet chamfers are removed from the model. The frame material is cast aluminum ZL201, and its material properties are shown in Table 1.

| Material name | Density (Kg/m3) | Poisson's ratio | Elastic modulus (Pa) |
|---------------|-----------------|-----------------|----------------------|
| ZL201         | 2700            | 0.33            | 7.1×10^10            |

The major force of the frame is the load carried by two axle holes in the pitch direction \( F \), so the stress of the load on the two axis holes in the pitch direction of the frame can be expressed as:

\[
\sigma = \frac{F}{2A}
\]  

In formula (1), \( A \) is the bearing area of each pitch axis hole. The strain \( \varepsilon \) of frame under load \( F \) can be expressed as:

\[
\varepsilon = \frac{\sigma}{E}
\]

In formula (2), \( E \) is elastic modulus of the frame material. The results of model pretreatment of the frame are shown in Figure 3. It can be seen from Figure 3 (a) that the frame is connected with the support frame through two shaft holes to realize rotation in the roll-direction, so that the two shaft holes in roll-direction are constrained.

4. Theoretical model of vibration

Considering the platform as an elastic body, its vibration equation under undamped free vibration is:

\[
[M] \cdot \ddot{Z}(t) + [K] \cdot Z(t) = 0
\]  

In equation (3), \([M]\) is the mass matrix of the platform, \([K]\) is the stiffness matrix of the platform, \(\ddot{Z}(t)\) is acceleration of the platform, \(Z(t)\) is displacement vector of the platform.

Assuming that the free vibration of the platform is a superposition of a series of simple harmonic motions, then the solution of equation (3) can be expressed as:
\[ \{Z(t)\} = \{Z_0 \sin wt\} \]  

(4)

In formula (4), \(w\) is the resonance frequency of the platform. Substituting equation (4) is brought into equation (3), as in

\[ ([K] - w^2[M]) \cdot \{Z_0\} = 0 \]  

(5)

When the platform is freely vibrating, the amplitude \(Z_0\) of each point in its structure is not all 0, so there must be a coefficient matrix determinant equal to 0 in equation (2), that is

\[ \det([K] - w^2[M]) = 0 \]  

(6)

Equation (6) is the characteristic equation of the platform structure, and its characteristic values solution \(w_1^2, w_2^2, ..., w_n^2\) can be obtained. And \(n\) linearly independent \(n\)-dimensional feature column vectors corresponding to each feature value \(\varphi_1, \varphi_2, ..., \varphi_n\). In the modal analysis of the platform, the square root of the eigenvalue \(w_i (i = 1, 2, ..., n)\), For its structure \(i\) order natural frequency eigenvector \(\{\varphi_i\}_{i=1}^{n}\). For the platform \(i\) order natural mode, composed of various order modes \(n \times n\) square matrix \([\varphi]_{i=1}^{n}\). In the modal matrix of platform vibration, the frame material of the platform is a hard aluminum alloy with higher specific strength. The gyro, motor, slip ring, encoder and visible light camera are used as the rigid body. The density is converted according to the actual situation.

5. Calculation results and analysis

ANSYS software is used to calculate the engineering mechanics of the frame, and the statics and vibration modal analysis results of the frame under the loading state are obtained.

When the frame is fixed relative to the support frame, a load of 200kg is applied. Under 3g acceleration in gravity, the static analysis results of the frame are obtained, as shown in Figure 4. The stress distribution of the frame is shown in Figure 4 (a), with the maximum compressive stress of 5.18mpa and the maximum tensile stress of 8Mpa.

\[ \text{(a) Stress distribution} \quad \text{(b) Strain distribution} \]  

Figure 4. Results of frame static analysis

According to the allowable stress 295mpa of cast aluminum ZL201, the safety factor of frame structure is about 37. The strain distribution of the frame is shown in Figure 4 (b). The maximum deformation occurs at the shaft hole. The maximum compressive strain is 0.0558mm, and the maximum tensile strain is 0.046mm. On this basis, the structure of the frame is optimized, the weak
part of the stress axis hole of the frame is strengthened, the wall thickness of the stress part is increased, and the stiffener is added around the frame to improve its mechanical properties. The improved static analysis results are shown in Figure 5.

It can be seen from Figure 5 (a) that the stress distribution of the frame is under the same load and constraint, the maximum compressive stress of the optimized structure is 1.17mpa, the maximum tensile stress is 2.55mpa, and the safety factor of the frame structure is increased to 115, which shows that the rigidity of the optimized frame structure is more than twice that before optimization. As shown in Figure. 5 (b), the maximum compressive strain and tensile strain of the optimized frame are 0.015mm and 0.019mm respectively. It can be seen that the strain after optimization is reduced to about 1/3 of that before optimization, which shows that the deformation of the frame structure is significantly reduced, thus the control accuracy of the system is greatly improved.

![Stress distribution](image1)
![Strain distribution](image2)

**Figure 5. Statics analysis results of optimized frame**

Based on the analysis of the frame statics, the vibration modes of the frame are analyzed, and the resonance frequency of the frame is calculated from 0 to 2000 Hz. Figure 6 shows the first-order modal results before and after the frame optimization.

It can be seen from Fig. 6 (a) that the first-order resonance frequency obtained is 37.95Hz, which is relatively low. It can be seen from Fig. 6 (a) that the First-order resonance frequency of the optimized structure is 195.32Hz, which indicates that the improved resonance frequency has been greatly improved.

![Before optimization](image3)
![After optimization](image4)

**Figure 6. Comparison of modal analysis results before and after frame optimization**

The results of Fifth order modal analysis before and after frame optimization are shown in Table 2. From the data in Table 1, it can be seen that the 5th modal resonance frequency of the frame structure after optimization is greatly increased, which shows that the dynamic characteristics of the frame have been effectively improved.
Table 2 Comparison of 5-order modal results before and after frame optimization

| Modality Order | Before optimization | After optimization |
|----------------|---------------------|--------------------|
| Resonant frequency (Hz) | Mode description | Resonant frequency (Hz) | Mode description |
| First order | 37.95 | Rotate in Y direction | 195.32 | Rotate in Y direction |
| Second order | 56.58 | Swing in X direction | 229.73 | Swing in X direction |
| Third order | 84.15 | Swing in Z direction | 269.8 | Swing in Z direction |
| Fourth order | 269.82 | Rotate in Z direction | 347.6 | Rotate in Z direction |
| Fifth order | 284.62 | Rotate in X direction | 456.1 | Rotate in X direction |

6. Conclusion
Due to the particularity of the application environment of the platform, it is not only required to be light enough, but also to have high load capacity and stability accuracy, so there is a certain difficulty in the design of its frame structure. In the paper, the finite element method is used to analyze the static and modal characteristics of the structure, and the structure is optimized according to the analysis results. The results show that the stiffness of the optimized frame structure is greatly improved, and the dynamic characteristics are also effectively improved. Therefore, it is necessary to analyze the engineering mechanics of the stable platform. The analysis method in this paper can provide some theoretical basis and reference value for the design of AISP.

References
[1] Xue Dan. Discussion on the frame structure of photoelectric stabilized platform. Photoelectricity information. 2011, 28(3):33~36.
[2] Masten M K. Inertially stabilized platform for optical imaging system: tracking dynamic targets with mobile sensors[J]. IEEE Control Systems Magazine, 2008(12): 47~64.
[3] Hilker J M. Inertially stabilized platform technology: Concepts and principles[J]. IEEE Control Systems Magazine, 2008(12): 26~46.
[4] Yang Pengjun, Wang Jiamin. Finite element analysis of inertial platform components [J]. Mechanics and electronics, 2004(4):15~18.
[5] 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.
[6] Cheng Feng, Ren Ge. Modeling and vibration transfer rate analysis of inertial stabilization platform. Optoelectronic Engineering, 2006, 33(4):19~22.