Modal Analysis of Work Bench Gear Shaft Based on ANSYS

Sheng Chang[1, a], Jian Liu[1, b]

[1]Shandong University of science and technology, Shandong 266590 China.

[a]1447965727@qq.com, [b]2415201321@qq.com

Abstract

In order to study the vibration characteristics of gear shaft, the modal analysis was carried out by the engineering analysis software work bench. This paper describes the whole process of modal analysis in detail, from the selection of modeling environment, the establishment of gear shaft solid model and finite element model to the final modal analysis. Among them, the constraint is set to the surface where the gear is located, so as to ensure the gear to carry out its mode under normal operation. The 10th order natural frequency and formation of the shaft are obtained through analysis. Compared with the transfer matrix method, the method is simple and convenient, the calculation is quick, and a more intuitive and vivid formation is obtained. Modal analysis effectively predicts the vibration characteristics of the structure and provides a reference for the selection of shaft materials so as to better analyze and optimize the characteristics and lay a foundation for harmonic response analysis and transient analysis.

Keywords

Modal analysis; Natural frequency; Formation; vibration performance.

1. Introduction

The gear shaft is an important part of the reducer. The dynamic characteristics of the gear shaft under high-speed operation, such as vibration, noise and stability, are important indicators to judge its performance and also important factors to affect the overall performance of the reducer. Therefore, the modal analysis of the gear shaft can provide a theoretical basis for the correct and reasonable design of the gear shaft, making it have a good dynamic performance index, and solving the problem of structural dynamic performance defects. Aiming at the gear shaft of a certain type of reducer, this paper uses different software at different stages of modal analysis such as geometric modeling, mesh generation, analysis and calculation, and comprehensively uses the characteristics of SolidWorks software and ANSYS Workbench K software to improve the efficiency and accuracy of problem solving.

2. Modal Analysis Basis

Modal analysis is about finding eigenvalues and eigenvectors. Eigenvalues are about knowing the frequencies corresponding to some basic vibration modes of the structure. In practice, sometimes in order to avoid these basic frequencies and prevent resonance, and sometimes to strengthen vibration, depending on the actual needs, the basic natural frequencies can give us a criterion to know whether the deformation of our structure is fast or slow, and the basic natural frequencies can also represent the stiffness of the structure as a whole: a low frequency means that the stiffness of the structure is very low (the structure is very soft), while a high frequency means that the stiffness of the structure is very high (the structure is very hard). The degree of hardness of the structure varies according to the requirements. For example, rigid tall building design is not likely to shake too much, but it is not easy to absorb seismic energy. On the contrary, the flexible tall building design will shake a lot, but it can often absorb a lot of seismic energy.

Modal analysis is used to determine the vibration characteristics of the designed structure or machine components, i.e. the natural frequencies and vibration modes of the structure, which are important parameters in structural design under dynamic loads and can also be used as a starting point for other dynamic analysis problems. ANSYS Work Bench can perform modal analysis on prestressed structures and circularly symmetric structures. Gear shaft is an important part of the machine, so modal analysis of gear shaft is of great significance.

Modal analysis with viscous damping is a classical problem. For a single-degree-of-freedom system, it is not difficult to write the differential equation of the system using Newton's second law or Lagrange's equation:

$$\mathrm{m}\ddot{x} + c\dot{x} + kx = 0 \tag{1}$$

Where m is a measure of the inertia of the rigid body's linear motion, called mass, with a range of kg;C is called damping coefficient and the dimension is ns / m; K is the spring stiffness and the dimension is KGM.

According to the theory of solving differential equations, the general solution of equation (1) has a form

$$\mathrm{x} = \mathrm{A}e^{\lambda t} \tag{2}$$

The characteristic equation of the system obtained by substituting (2) into (1) is:

$$\mathrm{m}\lambda^2 + c\lambda + k = 0 \tag{3}$$

Solve the characteristic equation (3) to obtain the characteristic value as follows

$$\lambda_{1,\ 2} = -\frac{c}{2m} \pm \sqrt{\frac{c^2}{4m^2} - \frac{k}{m}} \tag{4}$$

It can be seen from equation (4) that the damping coefficient c has an influence on the characteristic value. When c changes, the root expression at the right end of equation (4) may be greater than zero, equal to zero, or less than zero, corresponding to the appearance of two different real roots, heavy roots, or conjugate roots respectively, so the corresponding vibration response characteristics are also different.

In formula (4), the damping coefficient c with a root value of zero is defined as the critical damping coefficient c_c, is

$$c_c = 2\sqrt{mk} = 2m\omega_n \tag{5}$$

In order to facilitate the analysis of damping parameters, the following damping ratios or damping factors ζ are introduced, namely

$$\zeta = \frac{c}{c_c} = \frac{c}{2\sqrt{mk}} \tag{6}$$

Substituting equation (6) into equation (4), the eigenvalue expressed by dimensionless damping ratio ζ is obtained as follows

$$\lambda_{1,\ 2} = -\zeta\omega_n \pm \omega_n\sqrt{\zeta^2 - 1} \tag{7}$$

Obviously, when the damping ratio ζ is different and the eigenvalues are different, the system will generate different motions.

For both critical damping and overdamped states, the corresponding motion is non-oscillatory damping motion and does not have vibration characteristics, so we will only discuss the underdamped state ($\zeta < 1$) here. The response in the underdamped state is the product of two factors, one is an exponential function decaying over time and the other is a sinusoidal function, so the overall response is a quasi-periodic vibration with exponential attenuation of amplitude, rather than a strictly periodic vibration [2]. As shown in fig. 1:

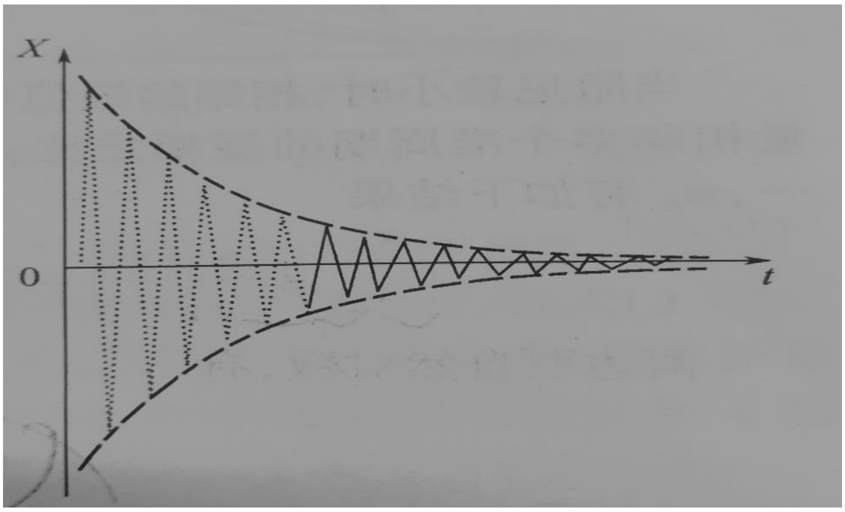


Figure 1. Underdamped Case (ζ <1)

3. Analysis Process

3.1 Establish Entity Models

Before the finite element analysis, it is necessary to build a solid model, and CAD modeling can be used for less complicated parts. SolidWorks is used to build the model, and its format is IGS. The model is imported into ANSYS through the Work Bench interface. The model is shown in Figure 2.

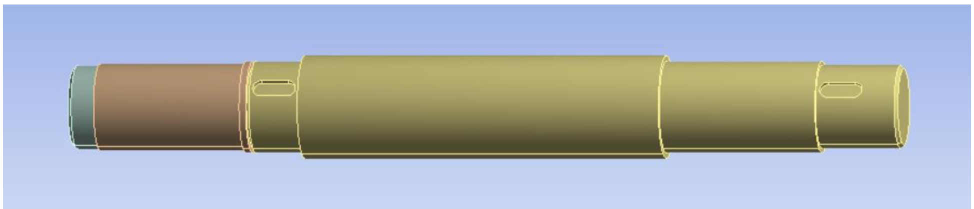


Figure 2. Entity Model

3.2 Establish Analysis Project

First, you need to set the model unit and click Modal to create an analysis project, as shown in Figure 3.

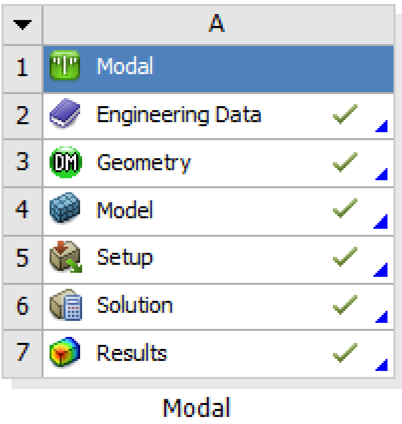


Figure 3. Analysis Project

3.3 Material Parameter Setting

This shaft rotates around the center. First, we need to import the engineering data source and set the material as stainless steel alloy. The material parameters of the shaft are: Young's modulus 200×10^9Pa, densityρ=7850kg/m^3, Poisson's ratio 0.25.

3.4 Grid Division

Grid division will have a direct impact on calculation accuracy and calculation scale. In static analysis, if only the deformation of the structure is calculated, the number of grids can be smaller. If stress needs to be calculated, a relatively large number of grids should be taken under the same accuracy requirement. Also in response calculation, the number of grids used to calculate stress response should be larger than that used to calculate displacement response. When calculating the inherent dynamic characteristics of the structure, if only a few low-order modes are calculated, fewer grids can be selected, and if the calculated mode order is higher, more grids should be selected. In order to better reflect the law of data change, a relatively dense grid is needed in places where the calculated data change gradient is large (such as stress concentration). In order to reduce the scale of the model, a relatively sparse grid should be divided where the change gradient of the calculated data is small. In this way, the whole structure shows different grid division forms. The grid division is shown in Figure 4.

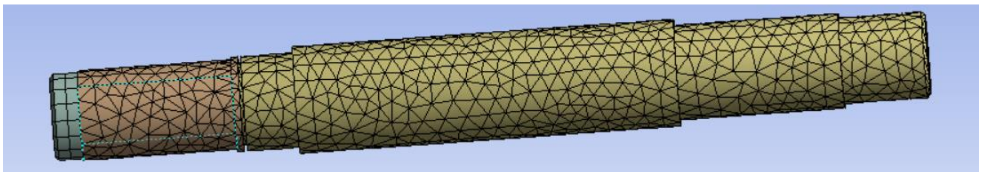
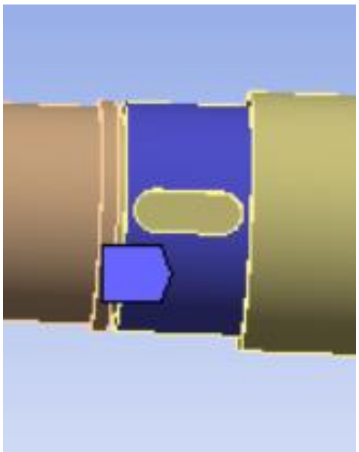


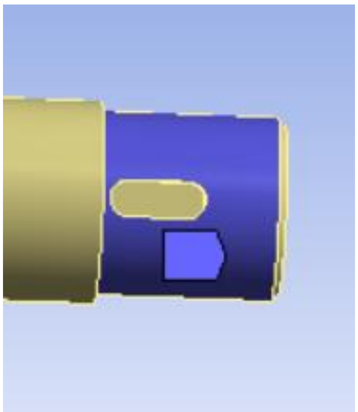
Figure 4. Model after Grid Division

3.5 Applied Load and Boundary Conditions

It is necessary to apply a fixed constraint on the part where the gear is mounted to ensure that the gear is not affected by shaft deformation. the fixed constraint application part is shown in fig. 5



Fixed Support1



Fixed Support2

Fig 5. Application of fixed constraints

4. Set Up Solution and Modal Analysis

The model is divided into blocks by LANC ZOS algorithm, which is a powerful method. This method is very effective when extracting a large number of vibration modes from medium to large models (50 to 100 degrees of freedom). It is often used in models with solid elements or shell elements. The rigid vibration mode can be well handled. The vibration of the structure can be expressed by linear combination of natural vibration modes of each order, among which the lower natural vibration modes have greater influence on the vibration of the structure than the higher natural vibration modes, and the lower vibration modes play a decisive role in the dynamic characteristics of the structure [3]. In the modal analysis of eccentric shaft, the first 12 frequencies of the shaft are extracted, and then the first 12 frequencies of the shaft are calculated by ANSYS, and the data are derived to obtain the data of each frequency as shown in Table 1.

Table 1. First 12 Frequency Data of Gear Shaft

	Mode	☑ Frequency [Hz]
1	1.	1750.2
2	2.	2032.
3	3.	2105.9
4	4.	2227.9
5	5.	4342.9
6	6.	4566.7
7	7.	5004.9
8	8.	6488.5
9	9.	6734.4
10	10.	8235.2
11	11.	8743.1
12	12.	9038.

According to the natural frequency obtained by finite element simulation, the first few order critical rotational speed $n = 60\Omega$ of the spindle is calculated, and then according to the actual working condition of the gear shaft, the critical rotational speed close to the actual rotational speed at a certain order frequency is selected. The magnitude of the vibration mode is only a relative magnitude (displacement relative value), which represents the relative ratio of vibration magnitude of each point at a certain natural frequency, reflects the transmission of vibration at that natural frequency, and does not reflect the value of actual vibration [4]. The modal analysis results are shown in fig. 6.

From the shape of the vibration state, we can know the deformation trend of the structure at a certain natural resonance frequency. If you want to strengthen the rigidity of the structure, you can strengthen it from these weaker parts. The role of modal analysis is mainly reflected in: (1) providing basis for selecting motor parameters; (2) Increasing the rigidity of the main shaft bearing can effectively increase the critical speed of the eccentric shaft and avoid the resonance region; (3) The modal analysis is more accurate than the transfer matrix method in solving the natural frequencies and modes, and lays a foundation for harmonic response analysis and transient analysis.

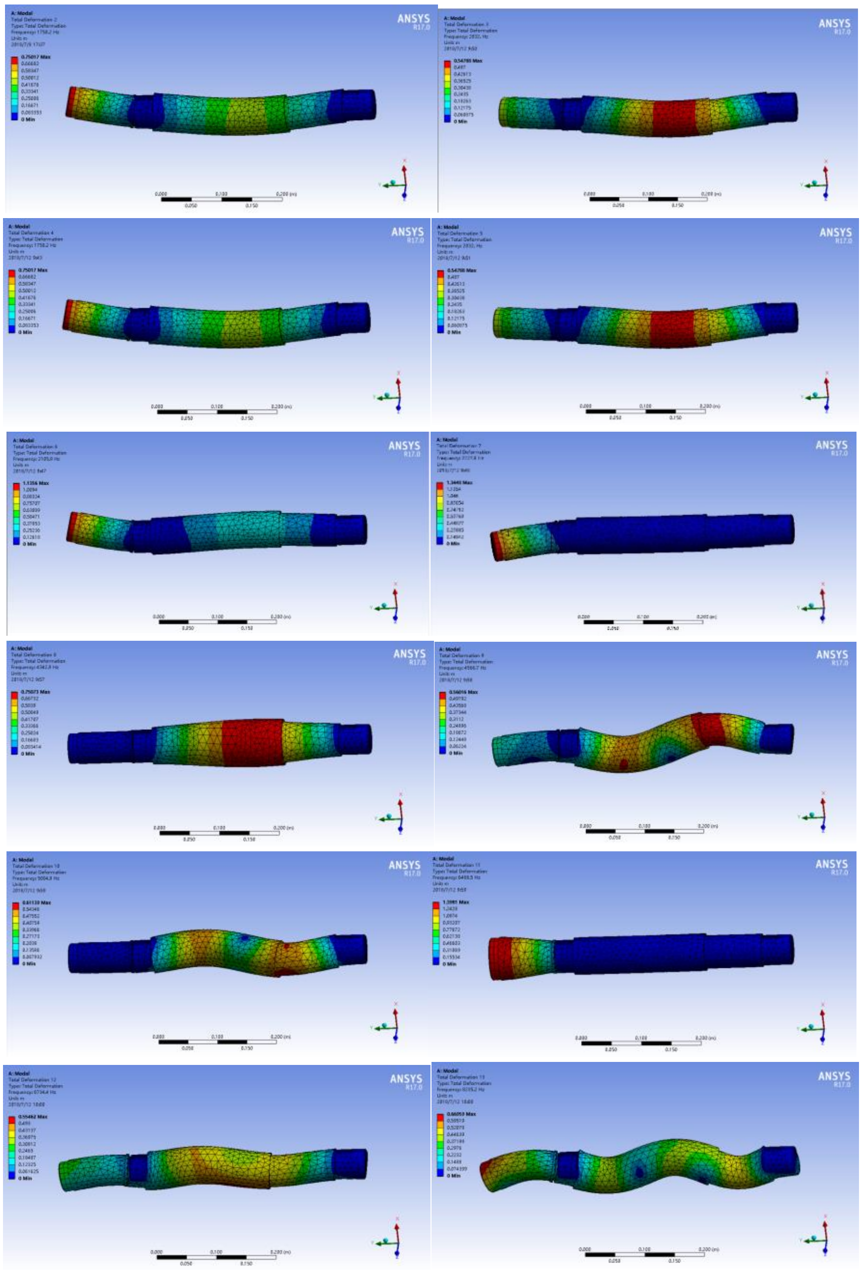


Fig 6. Vibration mode diagram at 12th order frequency

5. Conclusion

(1) The minimum frequency of the gear shaft is 1750.2 Hz, and the highest frequency of the first 12 steps is 9038 Hz. Therefore, the resonance phenomenon is inevitable when the shaft starts to rotate.

(2) The corresponding mode shapes at the 1st to 6th order frequencies are favorable for us to qualitatively investigate the deformation of the shaft because the corresponding rotational speed at these frequencies is close to the actual rotational speed.

References

[1] Liu Zuoshi, Peng Jianyun. Modal Analysis of Eccentric Shaft of Vibrating Screen Based on ANSYS [D]. Computer Application Technology, 2010, (2).

[2] Wang Fumao, Ren Yongsheng, Han Baokun. Fundamentals of Mechanical Vibration and Noise Analysis [M]. Beijing: National Defense Industry Press, 2015: 17 - 21.

[3] Sun Aiqin, Wang Jidai. Dynamic Analysis of Motor Spindle of High Voltage Inspection Robot Based on ANSYS [J]. Coal Mine Machinery, 2008, (7): 83 - 84.

[4] Yuan Anfu, Chen Jun. Application of ANSYS in Modal Analysis [J]. Manufacturing Technology and Machine Tools2007, (8): 79-82.

[5] Cao J, Krishnan N, Wang Z, Lu H, Liu W K, Swan-son A. Microforming: Experimental investigation of the extrusion Process for micropins and its numerical simulation using RKEM [J].Journal of Manufacturing Science and Engineering, 2004, 126: 642-645.

[6] SchuberA, BurkhardtT, NeugebauerJ, etal. High Precision Embossing of Metallic PartswithMicor structure [C]. The lstlntenrational Eus Pen Coneferencc, Aachen, Shaker, 1999.