Static Characteristics Analysis of ZK Worm and Worm Gear Based on ANSYS Workbench

Xu Yun [a], Yuankai Meng [b], Yinghua Liao [c], Yan Shi [d]

School of Mechanical Engineering, Sichuan University of science & engineering, Zigong, 643000, China.

[a]1214004131@qq.com, [b]450160884@qq.com, [c]191151820@qq.com, [d]408624969@qq.com

Abstract

In order to check the static stiffness of the ZK worm and worm gear, this paper analysis the static characteristics of the ZK worm and worm gear of the indexing slewing device with the finite element analysis software ANSYS Workbench. In this paper, the model is simplified reasonably, and the finite element model is established in ANSYS. The static deformation and static stiffness of the ZK worm and worm gear are obtained through analysis and calculation. The results show that: the static stiffness of the ZK worm and worm gear can meet the design demand; The finite element modeling method is accurate and effective, meanwhile the analysis method can also be applied to static and dynamic analysis of other key components of the indexing slewing device.

Keywords

ZK worm and worm gear; ANSYS Workbench; Static characteristics; Finite element analysis.

1. Introduction

The machine indexing mechanism is mainly used to realize the circumferential stepping (indexing) movement of the mechanical rotating components, after the indexing is finished, the mechanical rotating components is fixed at the determined position by the location device. Generally, the indexing mechanism itself has a location device, but the positioning accuracy is high and the original device cannot be reached, or the transmission system is used to set the location device. The indexing slewing device is widely used in applications requiring accurate transmission motion because of its advantages of steady transmission, constant transmission ratio, compact structure, and ability to withstand large loads. The indexing slewing device must have a good static characteristic to ensure the reliability and positioning accuracy. However, the ZK worm and worm gear is the core component of the indexing slewing device, its static stiffness directly affects the positioning accuracy of the indexing slewing device.

The main failure mode of the worm drive is the tooth flank fatigue pitting and scuffing, both of which are related to the load distribution on the contact area of the worm pair. It is always a concern that the contact strength of the tooth surface of the worm pair is analyzed by an effective method. With the rapid development of computer technology, the finite element analysis has become an effective reference method for solving complex field problems with its own advantages. In the field of mechanical transmission, there are many researches on the simulation analysis of gear transmission using finite element meth. For example, Yi-Cheng Chen [1] used the finite element analysis (FEA) method to study the contact stress and the bend stress of the gear pair with partial bearing contact. Tengjiao Lin [2] a three-dimensional static and three-dimensional finite element method for dynamic contact/impact analysis of gear transmission, Kousaku Ohno [3] three-dimensional finite element method analysis of the contact stress of helical gear and He Yunhua [4] three-dimensional parametric finite element analysis of the helical gear strength, but less research on finite element simulation analysis of worm drive. Therefore, based on the three-dimensional model of the ZK worm and worm gear, an attempt was made to study the three-dimensional static characteristics.

In this paper, the contact problem of the worm drive is one of the most common contact problems in mechanical engineering. Traditionally, the contact strength of the worm and worm gear is calculated that based on the contact strength of the gear transmission, to the actual problem is simplified. In this paper, the contact strength analysis of ZK worm and worm gear is realized by finite element method on ANSYS software.

2. A simplification of the analytical model

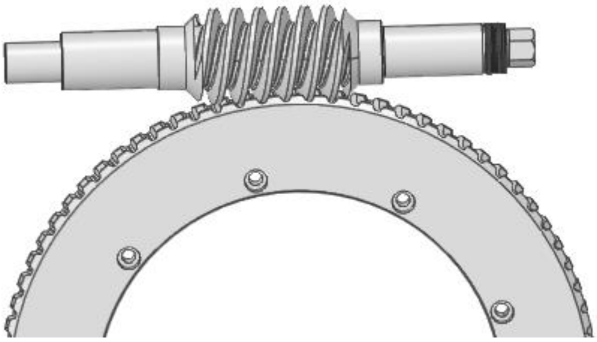

Figure. 1 Simplified model of ZK worm and worm gear

Taking into account the computing power of computer software, under the condition of ensuring the accuracy and reasonableness of the solution results, the following assumptions or conditions restriction is proposed to slove model of the FEA model of the ZK worm and worm gear.

(1) Taking into account the efficiency of the calculation, according to the Saint-Venant's Principle and the periodicity of the gear meshing, the entire region is reduced to the gear teeth directly involved in the meshing, and the solution domain range and the boundary displacement constraint are reasonably determined.

(2) The contact surface is continuous and smooth, regardless of temperature changes, and the friction acting on the contact surface follows the Coulomb friction rule.

(3) The material of the object is continuous, linearly elastic and uniform.

According to the above assumptions, the simplified FEA analysis model of ZK worm and worm gear is shown in Figure 1.

3. Simulation Analysis of ZK Worm Gear Contact

3.1 Establishment of mechanical model of ZK worm and worm gear

The model diagram of the worm and worm gear analysis is shown in Figure 2, and the torque on the worm gear in Figure 2 M=5456248.85Nmm.

3.2 Import models and select materials

The Static Structural module is called in the Workbench for structural static analysis and then imported into the intermediate model file in stp format. Copper alloy and structural steel are added to the material library, and the mechanical properties of the material such as Density, Modulus of Elasticity and Poisson's Ratio are changed according to the material properties. Table 1 shows the relevant attribute of materials used for the worm, worm gear and spindle. In the specific analysis, the data of this table will be used to set the parameters of the material to better simulate the real situation and make the analysis result more reliable.

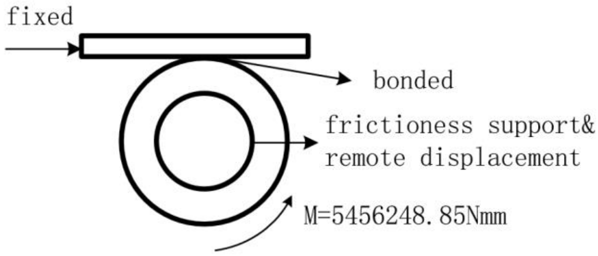

Figure. 2 Mechanics model of ZK worm and worm gear

Table 1 Material of worm-gear pair

Name	Material	Density ρ/(Kg·m3)	Elasticity modulus E/1011Pa	Poisson ratio μ
Worm	20CrSnTi	7800	2.07	0.25
Worm gear	ZQSn10-1	8800	1.24	0.3

3.3 Defining contact

For the contact analysis of the worm and worm gear transmission, since the surface stiffness of the worm gear teeth is larger than the worm wheel tooth, it is determined that the worm gear tooth surface is the target surface, and the worm gear tooth surface is the contact surface. After importing the model, Workbench will automatically identify the contact location and define the contact method, but still need to check whether the contact is reasonable and adjust. After the tooth surface contact is selected, the contact type is Bonded, so that it is not separated in the normal direction and the tangential direction, and the symmetric behavior Symmetric is selected at the Behavior. The algorithm formula selects Augmented Lagrange, because of its good characteristics and flexibility, and the enhanced Lagrangian formula adds additional control to automatically reduce penetration.

3.4 Mesh generation

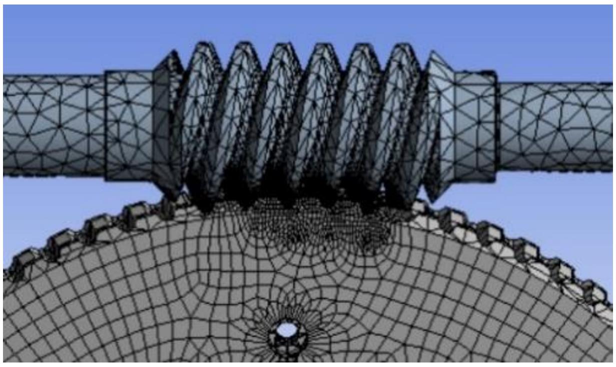

Figure. 3 ZK worm and worm gear wheel meshing mode

First observe the mesh divided by default, adjust the Relevance Center from Coarse to Medium, and the mesh will become denser. Since the emphatic focusing is the force condition of the contact position, the contact position is separately meshed. Insert Contact Sizing in the Mesh and enter 2.5mm in the Element Sizing. Finally, the number of meshes is 114648, and the number of nodes is 115406. The generated mesh is shown in Figure 3.

3.5 Set boundary conditions

According to the movement form of the worm drive in the indexing slewing device, the worm gear active and the worm wheel are driven, and a fixed constraint is applied to one end of the worm gear, and a Remote Displacement constraint is applied to the inner surface of the worm wheel to limit three translational degrees of freedom, and two rotational degrees of freedom, leaving only a Z-axis rotational degree of freedom. Simultaneous application of Frictionless Support without friction constraints limits the change in the normal. A Moment torque is applied to one end of the worm wheel, and the input value is "5456248.85 N/mm". After the torque direction is adjusted, the contact surface of the worm wheel and the contact surface of the worm gear when the contact is added are a pressed condition.

4. Finite element analysis result

After simulation calculation, from the analysis results of Figure. 4, the maximum equivalent stress is 169.5MPa, but the yield stress of the worm wheel material is 170MPa, so the worm wheel is qualified in terms of mechanical properties. In addition, it is concerned that the maximum equivalent strain is only 0.0018254 in Figure. 5,so the influence is negligible.

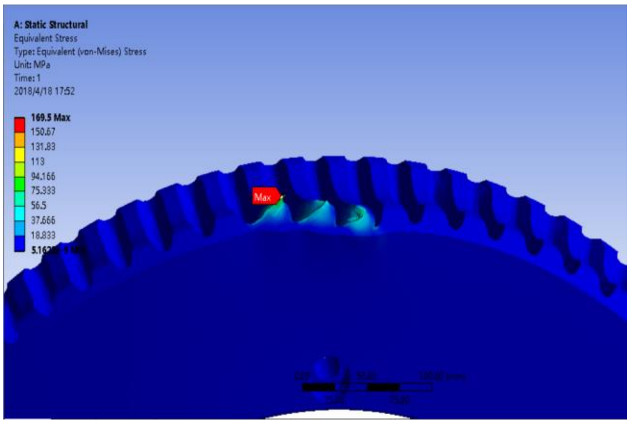

Figure. 4 ZK worm and worm gear equivalent stress map

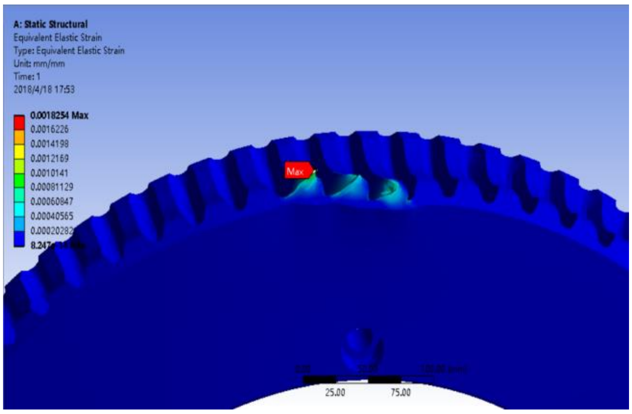

Figure. 5 ZK worm and worm gear equivalent strain cloud map

However, at the end of this analysis, the compressive stress of the worm and worm gear contact position needs to be understood. In the previous operation, the step of calling this result is not performed, but this does not affect the analysis. The calculation results performed in the Workbench platform are already in the background, the results only need to call and update the calculation on the interactive interface. When updating computer software, the computer will simply calculate some simple content without calculating large amount of data as the previous.

Then, the obtained compressive stress result (saving the step of the call result here) is compared with the allowable compressive stress to determine whether the worm wheel is qualified. The formula applicable to the allowable compressive stress of plastic materials as follows:

$$[\beta] = (1.5\sim2.5) * [\sigma] \quad (1)$$

According to the data in Table 1 above, $[\sigma]$ the allowable stress is obtained. According to equation (1, the allowable compressive stress calculated is 255 to 425 MPa. As shown in Figure 6 below, the compressive stress by the analysis and calculate is 150.02 MPa lower than the minimum allowable compressive stress of 255 MPa, so there is no problem at all, and the worm wheel is qualified.

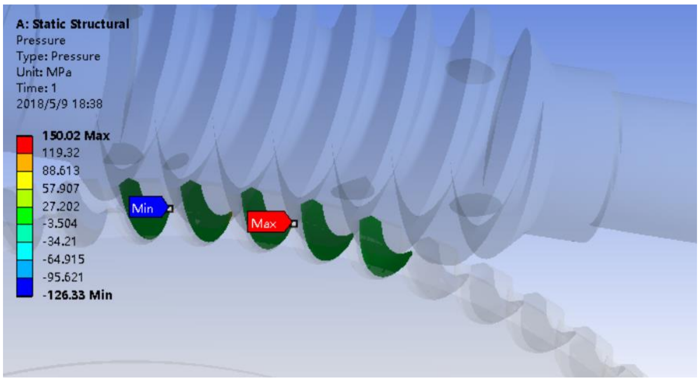

Figure. 6 Worm and worm gear contact position compressive stress cloud

5. Conclusion

The ZK worm and worm gear is a key component of the indexing slewing device, its static stiffness directly affects the indexing accuracy. It is very important to analyze the static characteristics of ZK worm and worm gear during the design process. It can be verified whether the static stiffness of ZK worm and worm gear meets the design requirements by finite element analysis. If it does not meet the requirements, it can be improved static characteristics of the ZK worm and worm gear by changing materials and transmission ratio.

In this paper, the finite element software is used to build the ZK worm and worm gear model. The static stiffness of the worm and worm gear is studied by static analysis. The finite element model of the ZK worm and worm gear is established with high accuracy. The maximum equivalent stress of the worm wheel satisfies the yield stress of the theoretically value. the compressive stress calculated by ANSYS is less than the minimum allowable value. So the worm and worm gear is qualified in terms of mechanical properties, which lays a solid foundation for the analysis of the dynamic characteristics of the indexing slewing device. The modeling and analysis method can also be applied to the static and dynamic characteristics analysis of other key components of the indexing slewing device.

Acknowledgements

This paper was supported by the Education Office of Sichuan Province (18TD0029), the Science & Technology Department of Sichuan Province (2017JY0148), the Science and Engineering of Science University (2016RCL01, y2017024), the Sichuan Provincial Key Lab of Process Equipment and Control (GK201817).

References

[1] Yi-Cheng Chen, Chung-Biau Tsay. Stress Analysis of a Helical Gear Set withLocalized Bearing Contact[J]. Finite Elements in Analysis and Design, 2002, (38): 707-723.

[2] Tengjiao Lin, H. Ou, Runfang Li. A Finite Element Method for 3D Static and Dynamic Contact/Impact Analysis of Gear Drives[J]. Comput. Methods Appl. Mech. Engrg., 2007, (196): 1716-1728.

[3] Kousaku Ohno, Naoyuki Tanak. A Contact Stress Analysis for Helical Gear with 3 Dimensional Finite Element Method[C]. Transactions of the Japan Society of Mechanical Engineers, 1998, 64(628): 4821-4826.

[4] He Yunhua. Three-dimensional parametric finite element analysis of helical gear strength. Master's thesis of Shandong University. 2007.