Advances in Engineering Research (AER), volume 148
3rd Workshop on Advanced Research and Technology in Industry Applications (WARTIA 2017)

Modal Analysis of CNC Lathe's Spindle Based on Finite Element

Kong Jiali, Cheng Xuelian

School of Mechanical Engineering, Shandong Huayu University of Technology, Dezhou 253034, China

kongjialiskd@126.com

Keywords: spindle, modal analysis, finite element, CNC lathe.

Abstract: With the development of the industry, the requirement of CNC lathe increase very much. To design a lathe, it needs analysis and calculation of the major components, and dynamic performance of the lathe spindle is very important for the performance and accuracy of the lathe. First, modeling of spindle structure are built in this paper. Secondly, modal analysis of spindle are carried out by using ANSYS software. Simulations show that the stiffness of spindle meets the design requirements.

1. Introduction

Machine tool industry is the key industry of machinery manufacturing industry, to provide equipment for the manufacturing industry[1]. The machine tool spindle is the key components of the machine tool spindle, a chuck mounting the front end of the workpiece or tool directly involved in cutting performance, especially the low order natural frequency has a great influence on the performance of the machine tool[2]. Especially for CNC machining it requires both high precision and high efficiency requirements, it is necessary to carry out and finish machining process, but also some rough machining, therefore, put forward higher requirements on the static and dynamic characteristics of the spindle. In this paper, the finite element machine spindle based on ANSYS analysis, provide a reference for the design of the spindle.

2. Finite Element Modeling of Spindle

Choosing the best modeling method is the key to establish an accurate model.The paper uses 3D modeling software SOLIDWORKS to establish the structural model of spindle, and then it use the interface function of ANSYS to import into SOLIDWORKS to set up finite element model by ANSYS software.The main parts of the spindle structure are shown in Figure 1. Because of the complex of CAD model, it cannot generate CAE model. So it is necessary to simplify the model and modify the spindle. The basic principle is that it ignores all the features in CAD model including small chamfer, round holes, convex, and linear processing of bevel in the model, etc. The removal can not account for the overall structure of the small parts of dynamic characteristics effect, it is seen in Figure 2.

According to the characteristics of modeling and structural mechanics of the spindle system, the spindle utilize the SOLID187 elements, which own quadratic function of the displacement, and it suitable for irregular mesh model. The unit for 10 nodes, each node own three degrees of freedom, X, Y, Z direction angle. Figure 3 shows SOLID187 unit figure.

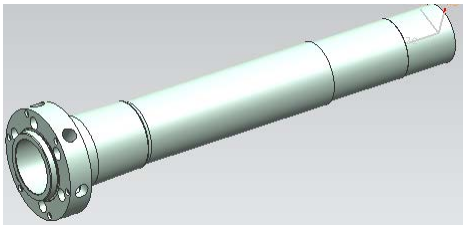


Figure 1 Structure of spindl

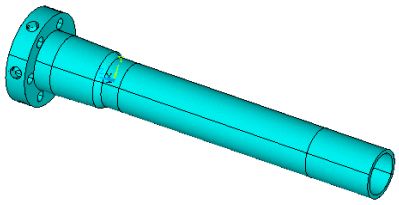


Figure 2 Schematic diagram of spindle

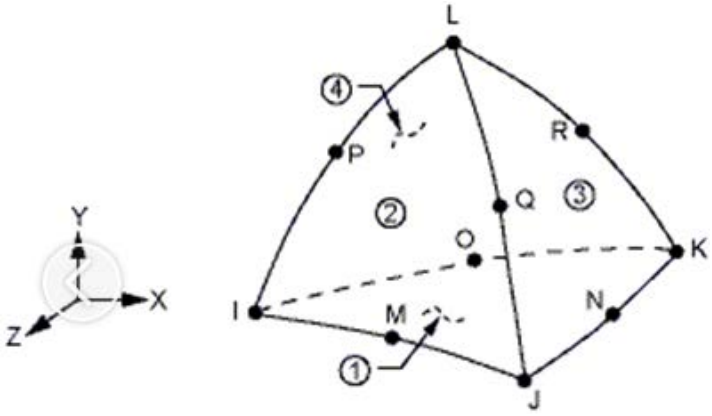


Figure 3 The SOLID187 unit

Table1 Material property

Modulus of elasticity $E(Pa)$	Density $\rho(kg/m^3)$	Poisson ratio μ
2×10^{11}	7.85	0.3

According to different parameters, ANSYS can achieve a variety of mesh, and mainly change complex model directly into the intelligent free mesh and grid local refinement, etc[2]. The CAD model are imported into ANSYS spindle, according to the actual boundary conditions, corresponding constraints on the model. The finite element model, with 103719 solid elements SOLID187, are shown in Figure 4.

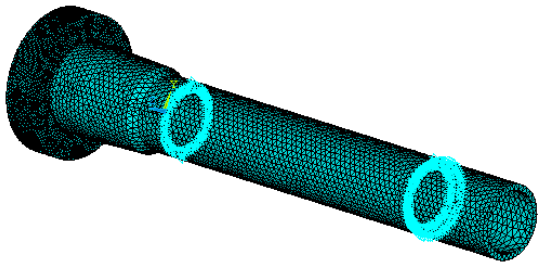


Figure 4 Finite element model of the spindle

3. Modal Analysis of Spindle Vibration

To know about the dynamic characteristics of spindle, the natural frequency and vibration mode are completed in the paper. According to the characteristics of the system, the dynamic analysis can divided into linear analysis and nonlinear analysis. According to the load change with time, it can be divided into dynamic analysis and transient dynamic analysis. ANSYS dynamic analysis provided the functions analysis of mechanics, on the basis of the needs of the spindle, modal analysis is carried out in the paper. The modal is inherent to the system structure and the vibration characteristics of the whole object[3]. Each mode own the specific frequencies and mode shapes, then modal analysis is a method to study the mechanics properties of the structure, which is also an important means of structure dynamic design and fault diagnosis. The main modal characteristics

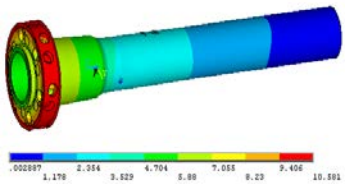
can master the structure system in a certain frequency range through modal analysis and predict the actual vibration in the external or internal response under various vibration sources.

ANSYS provides a variety of modal method, for example, block lanczos method, subspace method, power dynamics method, reduction method, asymmetric method,and damping method. This paper use the block lanczos method, which method is one of the most effective methods for solving large matrix eigenvalue problem, which is characterized by recursive vector matrix vector matrix formula to produce. Lanczos method is an orthogonal matrix multiplication operation, by using this method, it can get a hypothesis for the modal matrix of the discrete model of the structure of excellent quality. The low order modal space are formed, which can approximate the discrete model of the structure effectively[4].

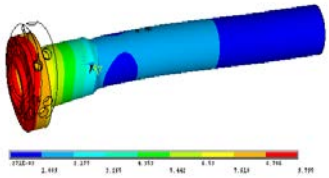
The vibration of the structure can be expressed as a linear superposition, including low order modes than the effect of higher modes of vibration on the structure.The dynamic characteristics of low order modes of structure play a decisive role, it analyzes the vibration characteristics of structure, which usually take 5-10 order. So this paper calculates the spindle of the first five order natural frequency and vibration mode, which can be seen in the Table 2 as follows.

Table 2 The 5 order natural vibration frequency

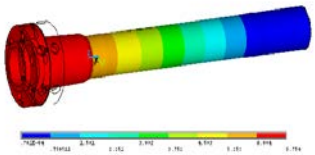
Order	Natural frequency value/ *Hz*
1	338.52
2	339.53
3	820.59
4	1043.9
5	1536.3



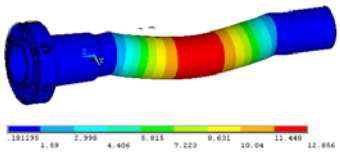
a)First order mode shapes



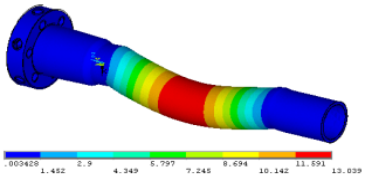
b)Two order modal shapes



c)Three order modal shapes



d)Four order modal shapes



e)Four order modal shapes

Figure 5 Diagrams of five modes

It can be seen from Table 2, that the natural frequency of the spindle is high enough, the spindle static stiffness can meet the design requirements of high stiffness. The first critical speed calculation of n=60×338.52=20311.2r/min according to the natural frequency of spindle. Modal analysis was done far greater than the working speed of spindle (less than 1600r/min), indicating that the work spindle speed can effectively avoid to ensure the machining precision of the spindle resonance.

From Figure 5, it can be seen that modal analysis can fast calculate. ANSYS mode are intuitive and easy to see from the modal analysis, the spindle's deformation give first place to bending, at the same time it own axial deformation. Therefore, the spindleis mainly in the bending deformation during work.

4. Conclusions

It is very important of analysis and calculation of the major components and dynamic performance of the spindle. This paper set up the modeling structure of the spindle at first. Then the modal analysis of spindle are completed by ANSYS software. Simulations results prove the stiffness of spindle satisfy to the design requirements.

References

[1] Liu H.W. (2010)Mechanics of materials. Higher Education Press,241-246.

[2] Woodford C.H. (1991).Engineering Analysis using PAFEC Finite Element Software. The finite element mesh,9,43-64.

[3] Bathe K. J.(1976) Finite Element Procedures in Engineering Analysis.Prentice.

[4] Fan X.W., Fan W. X. (2010)R6105 diesel engine crankshaft modal analysis of ANSYS. based on the mechanical design and manufacture, 11:37-38.

[5] Kong J. L. (2017) The design of simple slant bed horizontal CNC lathe. Tianjin University of Technology and Education, 32-39.