Computer Life (CPL) ISSN: 1819-4818
DELIVERING QUALITY SCIENCE TO THE WORLD

Statics Analysis of Finite Element for CNC Lathe of CK5250

Li Feng [a], Du XianWei [b], Shao Xiang [c]

Shandong University of Science and Technology, Qingdao 266590, China.

a414023666@qq.com, b65460213@qq.com, c760759321@qq.com

Abstract: This article use common finite element method, combining ANSYS Workbench with Solidworks. Three-dimensional solid finite element modeling and statics analysis of the lathe were carried out. It provides a powerful theoretical basis for the improvement of the structure and the optimization design in the future.
Key words: finite element, lathe, statics.

---

1. INTRODUCTION

Structural analysis can be divided into linear analysis and nonlinear analysis. The static analysis studied in this paper, the load is the linear load that is the weight of the structure and cutting force. CNC vertical lathe in the process of operation, the workpiece by the cutting force, the interaction between the force will pass through the various parts of the ground, due to the deformation be-tween the various components and thus cause the processing accuracy and surface quality. The precision of the machine tool is very high, which is why the machine tool has high precision and high efficiency. Therefore, it is necessary to grasp the design of a good machine tool through the statics analysis of the machine tool.

2. STATIC ANALYSIS

2.1 Static analysis of the methods and ideas

The accuracy and rationality of 3D modeling have great influence on the finite element analysis. The key components of the machine tool are castings, there are many fillets, steps and holes, in order to save the computer hour so as not to damage the computer, In the three dimensional modeling to simplify its processing, which omitted the process features casting roundness, transition fillet, chamfer, etc.; omit a variety of technology holes, threaded holes, positioning holes ,etc. CNC vertical lathe welding parts Set as the ideal welding component. The principle of simplification was verified and verified experimentally by Yang Yong liang of Dalian University of Technology when writing a master's thesis. The conclusion is that the chamfer, fillet and hole on the part, assembly or casting have little effect on the analysis, Can be ignored completely.

2.2 Static analysis of key components

In ANSYS Workbench, all the static analysis is done in the module "Mechanical", which includes the geometric model ---entity (the unit is 10 nodes tetrahedron-SOLID 187 and 20 nodes

hexahedron-SOLID 186); Material properties; contact and assembly type; environment loads and constraints; solution type and post processing.

3. STATIC ANALYSIS OF THE WHOLE MACHINE MODEL

In the statics analysis of the whole machine, In order to reduce the working hours of analysis and avoid the phenomenon of computer jam, the assembly of the whole machine includes only left and right columns, crossbeams, connecting beams, left and right tool holders and connectors.

ANSYS Workbench is used to mesh the whole machine, Solid1866, the default grid method, in which the "Details of Mesh" option opens the following "Sizing" item to set its parameters. Set the "Relevance center" option to "Fine"; set the "Span Angle Center" option to "Fine"; set the "Transition" option to "Slow"; set the "Smoothing" option to "High" to generate the grid. Then, check the quality of the grid: use the command get: Edit Mesh--Check Mesh. Edit Mesh Display Mesh quality. The number of units qualified in the grid is displayed by Standalone to show that the nodes of the automatic subdivision enter the domain of the adjacent cells without affecting the output of the results. Finally, penetrating is used to check the contact between assemblies. Fig. 1 shows the whole machine model after meshing.

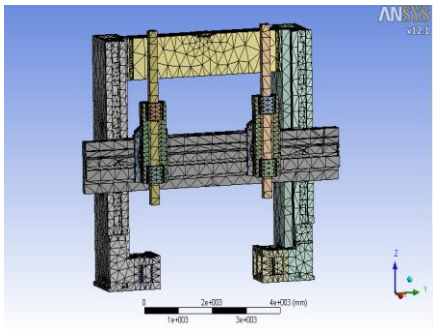

Fig. 1 The whole machine model after meshing

CNC vertical lathe is fixed by the anchor bolt at the bottom of the column. The side of the connector connected with the working base is fully constrained. The force on the vertical lathe is very complicated, and the forces and loads that make it deform include the gravity of the machine itself, the force of the workpiece in cutting and the weight of the workpiece being machined, The schematic diagram of the force is shown in Fig. 2.

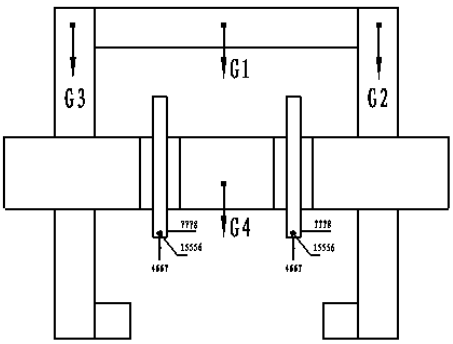

Fig. 2Force diagram of machine tool

There are two kinds of solvers: a---iterative solvers and b---direct solvers. The type of solver we determine is: a---iterative solver, then set up in the interface Tools-Options-Analysis Settings and Solution. Finally, the stress distribution diagram of the whole machine and the stress distribution of

the whole machine structure are obtained by loading and constraining in the X direction / Y direction / Z direction and the whole machine structure. As shown in Fig. 3 and Fig. 4

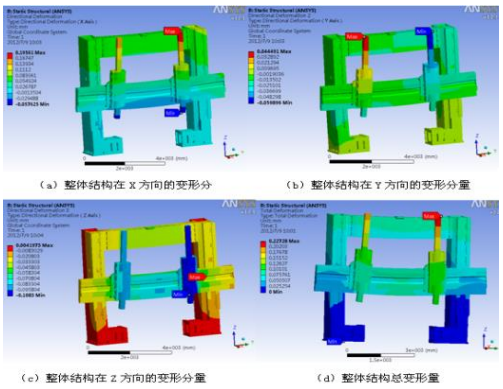

Fig. 3 Deformation of the whole structure in all directions

The deformation of X axis is 0.19 mm and the displacement of Y axis is 0.044 mm and the deformation displacement of Z axis is 0.004 mm. According to the manual of machine tool, when the precision grade is IT12 and the diameter of machining is 5000mm, the tolerance value is 2.1 mm, so the allowable deformation is 0.7mm. The maximum deformation of the X axis appears on the top of the sliding pillow and is not the key part of the machine tool, so the deformation of 0.19 mm is far less than 0.7 mm. The deformation of both Y-axis and Z-axis is smaller than that of finishing wheel, so the machining accuracy and position accuracy can be guaranteed.

The static stiffness of a double-column vertical lathe is the ability of the machine tool to resist the deformation of various parts under static load, because the machine tool itself is an elastic system. In addition to the deformation of the force, it will also cause the deformation of the contact surface of each part. Therefore, the greater the stiffness of the machine tool, the higher the dynamic accuracy of the machine.

When static stiffness is approximately linear, Static stiffness in the x direction $K_x = P_x / x$ ---the ratio of static load to comprehensive displacement in the x direction. Based on the above analysis results and the calculation formula of static stiffness, the stiffness of the machine tool in the direction of x, y, z is obtained as shown in Table 1.

Table 1 Results of statics analysis of whole structure

参数	X方向	Y方向	Z方向
作用力（N）	12222	3666	6111
最大变形（mm）	0. 1956	0. 0445	0. 0042
刚度（N. mm-1）	62. 4	82. 3	1456

According to the above calculation, the stiffness value is lower because of the large deformation in X and Y directions. This deformation is caused by the bending deformation of XOY plane, and in the deformation, the influence of the sliding pillow is great, so we want to improve the static stiffness of the machine tool. The stiffness of the sleepers should be improved to reduce the amount of deformation.

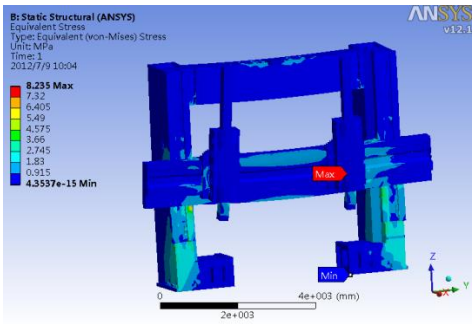

Fig. 4The stress distribution cloud diagram of the whole structure

According to the stress cloud diagram of the machine tool, the maximum stress is 8.235Mpa. the maximum value occurs at the position where the working point is concentrated at the turning tool. According to N= σ_Z / σ_{max} =230/8.235=27.9. Thus the maximum safety factor of the column is 27.9. The safety factor of the whole machine is very large and has a great margin to a certain extent. The overall structure can be optimized and the weight of the machine tool can be reduced.

4. CONCLUSION

In this paper, the static analysis of the sliding pillow, column, beam and the whole machine of the CNC vertical lathe is carried out by using the analysis software Ansys workbench12.1. The basic theory of the finite element method is introduced firstly. Then, the best import method between SOLIDWORKS and ANSYS workbench is studied. The key parts and the whole machine are meshed with the free mesh function of ANSYS workbench, and the main cutting force, feed force and back force are solved by empirical formula. Apply loads and constraints to the tool holder, finally The finite element analysis results of the key parts and the whole machine are tained.

It is concluded that all parts and components of the original machine tool and the whole machine have better stiffness and good stress distribution, but the safety factor is very large, which provides a theoretical basis for further improvement and optimization of the machine tool.

REFERENCES

[1] Jack C.H. Chung, Teng-Shang Hwang, Chien-Tai Wu, Yu Jiang, Jia-Yi Wang, Yong Bai, Hongliu Zou. Framework for integrated mechanical design automation [J]. Computer-Aided Design, 2000, 32(5).

[2] Zhang Y M, Liu Q W. Finite Element Analysis of Thermal Characteristics of the Spindle Assembly and Headstock for a Numerical Control Lathe [J]. Journal of Northeastern University, 2011, 32(4):571-574.

[3] Zhang Y M, Wang C Q, Lin X L. Finite Element Analysis and Optimization on the Numerical Control Lathe Bed[J]. Applied Mechanics & Materials, 2015, 778:28-32.

[4] Liu K, Zhong J, Liu G, et al. Dynamics analysis of superhard numerical control lathe based on ANSYS Workbench [J]. Journal of Beijing Information Science & Technology University, 2017.