Dynamic Analysis of Spindle-Column System in Finite Element Software

Mingxuan Du

School of Mechatronic Engineering, Southwest Petroleum University, Chengdu 610500, China;

kkjj1234@yeah.net

Abstract

In this research, a horizontal machining center was studied by finite element software to calculate modal frequencies and displacement response under excitation. Abaqus software was used for this research.

Keywords

Dynamic Analysis, Spindle-Column System.

1. Introduction

A horizontal center is consists of spindle, spindle box, column, lathe bed and stage. The object of this research is spindle-column system, which includes spindle, column and spindle box. Spindle is in horizontal direction, which can move with spindle box in column in vertical direction.

There are some scholars who have conducted some researches about machine tool, which involved many theories or methods, as shown in table below.

Table 1 Summary of theory or method in researches about machine tool

Researcher	Theory or method
Wu Wenjing [1]	Extended transfer matrix method
Liu Chengying [2]	Topology optimization and size optimization
Liu Haitao [3]	Perturbation theory
Yu Changliang [4]	Dynamic stiffness theory
Liu Linyan [5]	Joint interface theory
Qian Long [6]	Model simplification technique

Finite element method was used for this research.

2. Simulation

Simulation was conducted in finite element software Abaqus, which contains two parts: Modal analysis and Steady state analysis. Modal frequencies and modal shapes of spindle-column system can be calculated in Modal analysis step. The purpose of steady state dynamic analysis is to get displacement response of system under excitation.

2.1 Modeling

The models of spindle-column system were firstly established based on drawings of machining center in modeling software. In order to make simulation available in computer, some structures were removed and simplified, like some chamfers and holes. After modeling, these models were imported into Abaqus. Some geometries were broken because of importing, which needed to repair so that these models could be available in finite element software.

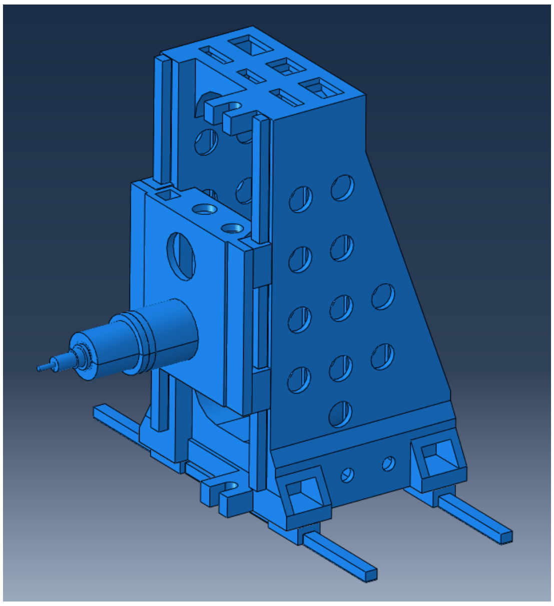

Fig. 1 Model of spindle-column system

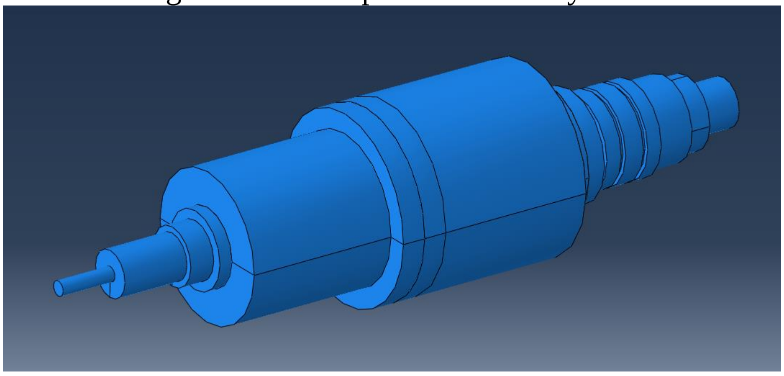

Fig. 2 Model of spindle

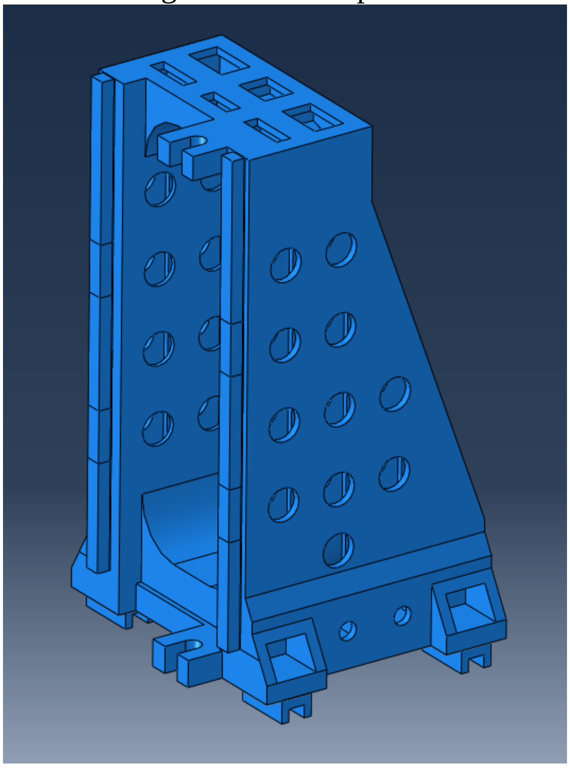

Fig. 3 Model of column

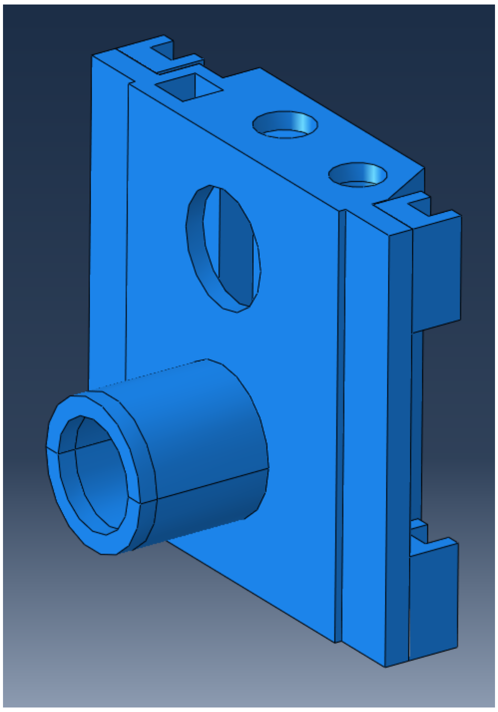

Fig. 4 Model of spindle box

2.2 Material properties

Different parts in spindle-column system are in different materials, which are main 4 different materials in spindle-column system, as shown in table below. Some other unimportant materials in spindle-column system are not considered.

Table 2 Material properties in simulation

Part	Material	Young's Modulus (MPa)	Poisson's ratio	Density (T/mm3)
Spindle	20Cr2MnMo	2.06E+05	0.3	7.87E-09
Spindle sleeve	45	2.10E+05	0.31	7.85E-09
Spindle box	HT250	1.25E+05	0.25	7.00E-09
Column	HT300	1.30E+05	0.25	7.30E-09

2.3 Constraint

There are some joints in machine tool, like bolt joint, ball screw joint, bearing joint and linear guide joint. In this simulation, linear guide joint and bearing joints were defined, which is responsible for linking different parts. There are 2 linear guide joints in this system. The first linear guide joint is between spindle box and column, and the second one is between column and lathe bed. Both of two linear guide joints were connected by "Tie" constraint in Abaqus. The slider was defined as slave surface and the guide was defined as master surface. Linear guide in bottom of column was fixed in lathe bed, so all the 6 DOFs of it were constrained. The bearing joint is between spindle and spindle sleeve, which was described by "spring" in simulation. Four springs were defined for one bearing in radial direction.

2.4 Element

Two kinds of elements were employed, which are 8-node linear brick, reduced integration, hourglass control elements (C3D8R) and 10-node quadratic tetrahedron element (C3D10). C3D10 element is suitable for almost any model, so it was used for main parts of spindle-column system, including spindle, column, and spindle box.

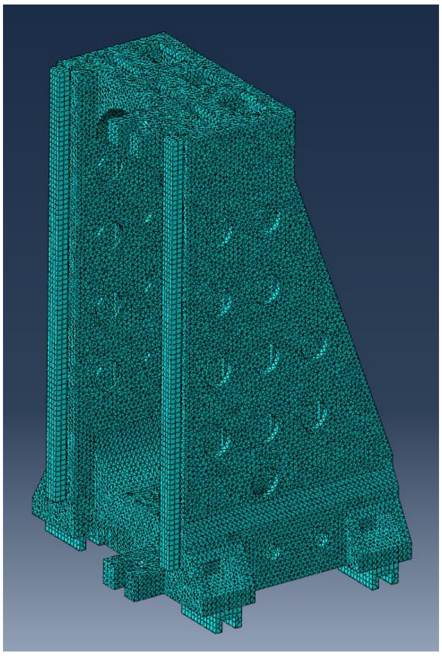

Fig. 5 Column model with mesh

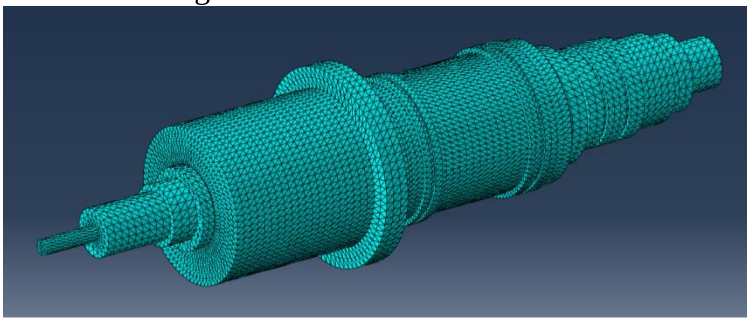

Fig. 6 Spindle model with mesh

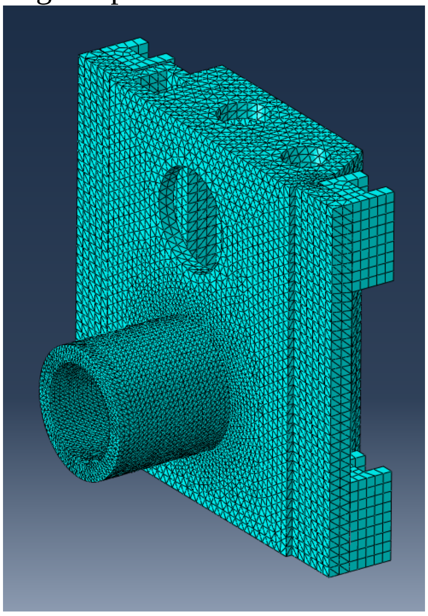

Fig. 7 Spindle box with mesh

The mesh size in spindle and spindle sleeve is smaller than those in spindle box and column because of its complexity. C3D8R elements were employed for slider and liner guide since it is easier for nodes in different parts to stay in same position. The number of elements in model is around 600 thousands.

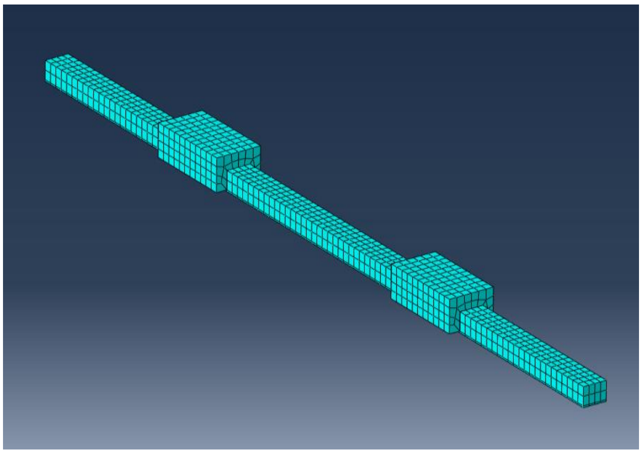

Fig. 8 Linear guide and slider with mesh

2.5 Modal analysis

There are 3 solvers in Abaqus for calculating modal frequency, which are Lanczos method, Subspace method and AMS method. Lanczos method was chosen for this simulation. The step was “Frequency” in “Linear perturbation”. First 30 modal frequencies and modal shapes were obtained, but only first 10 results were shown in paper.

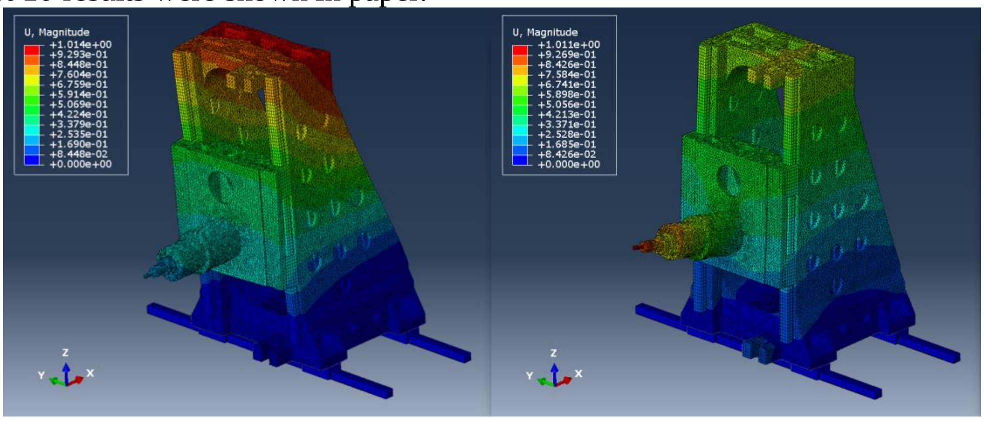

Fig. 9 Modal shape 1 and 2

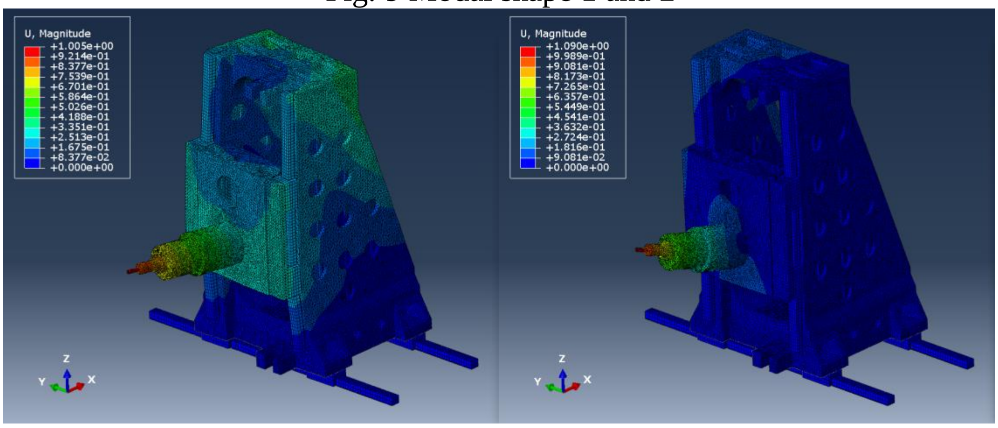

Fig. 10 Modal shape 3 and 4

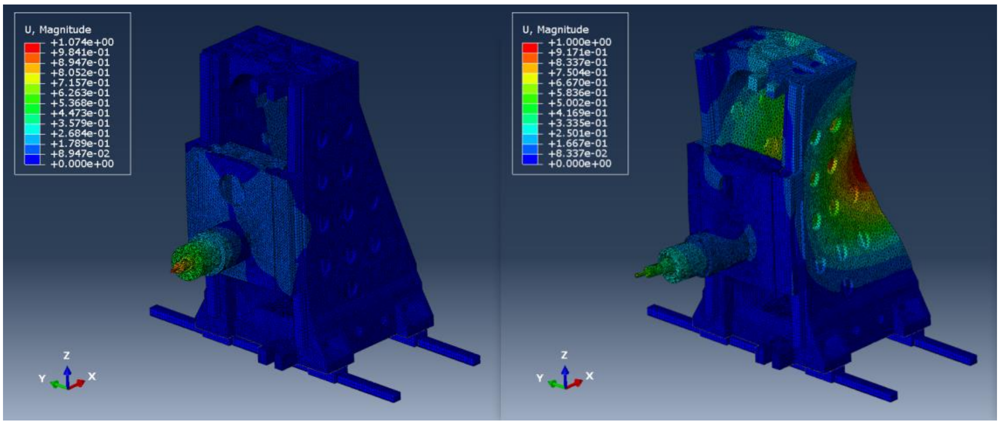

Fig. 11 Modal shape 5 and 6

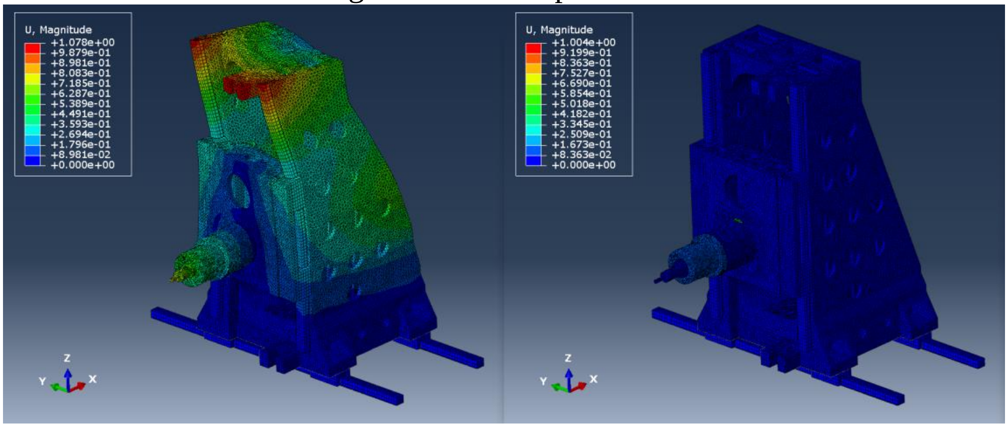

Fig. 12 Modal shape 7 and 8

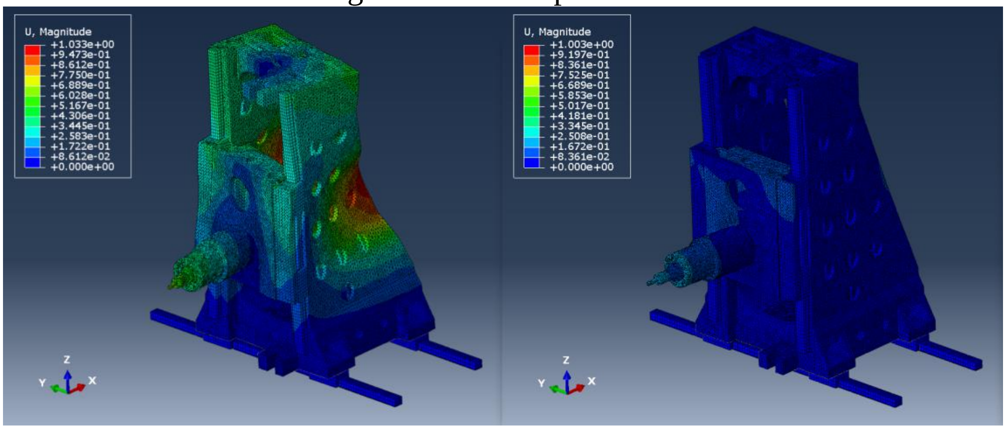

Fig. 13 Modal shape 9 and 10

Table 3 First 10 modal frequency from modal analysis

Modal number	1	2	3	4	5	6	7	8	9	10
Frequency (Hz)	87.681	146.81	165.23	220.86	241.87	257.71	268.23	311.83	336.53	347.95

2.6 Steady state dynamic analysis

After modal analysis, steady state dynamic analysis was conducted to get response under excitation. The harmonic force with 1 N amplitude was used for exciting force, which was applied to front end of spindle, as shown below. The frequency range was from 10 Hz to 900 Hz with 5 Hz increment, including mode 1 to mode 30. Damping constant was considered in this step, which is 0.005. The displacement in Y direction was extracted from another side of spindle. Figure below showed magnitude of displacement versus frequency.

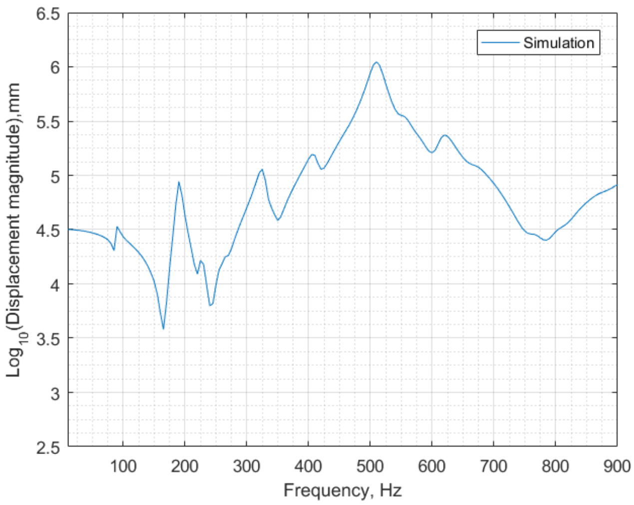

Fig. 14 Displacement magnitude versus frequency

As shown in figure above, the curve reached peak value in some modal frequencies from modal analysis step, like in frequency=85 Hz, 165Hz, 240Hz.

3. Conclusion

The maximum displacement is in around 500 Hz. Large number of elements in finite element model will cost much time to calculate, so work station is needed. The future work is to conduct hammer test in the same condition to get result to compare with Steady state dynamic analysis results.

Acknowledgements

The author would like to acknowledge the support of Southwest Petroleum University for this research.

References

[1] W.J. Wu, Q. Liu. Extended transfer matrix method for dynamic modeling of machine tools, Journal of Mechanical Engineering, vol. 46 (2010), 69-75.

[2] C.Y. Liu, F. Tan, L.P. Wang, et al. Research on optimization of column structure design for dynamic performance of machine tool, Journal of Mechanical Engineering, vol. 52 (2016), 161-168.

[3] H.T. Liu, W.H. Zhao. Perturbation analysis and optimization design for machine tool based on joint interface, Journal of Xi'an Jiaotong University, vol. 44 (2010), 96-99.

[4] C.L. Yu, H. Zhang, R.C. Wang, et al. Study on method for weak link identification of dynamic stiffness of a machine tool and optimization design, Journal of Mechanical Engineering, vol. 49 (2013), 11-17.

[5] L.Y. Liu, H.F. Wang, T.T. Liu, et al. Research on machine tool accuracy design system based on joints characteristics, Machine Tool & Hydraulics, vol. 40 (2012), 1-6.

[6] L. Qian, P.F. Feng, J.F. Zhang, et al. Research on model simplification procedure in the whole machine tool characteristic simulation analysis, Manufacturing Technology & Machine Tool, no. 9 (2016), 71-76.