Academic Journal of Science and Technology
ISSN: 2771-3032 | Vol. 4, No. 1, 2022

Analysis of Static Dynamic Characteristics and Structural Optimization of New Structural Vertical Plus Beams

Hao Zhang, Yun Xu, Jianjun Tao, Mingyue Yang, Ting Chen

School of Mechanical Engineering, Sichuan University of Science and Engineering, Yibin 644000, China

Abstract: As an important moving element in the machine structure, the crossbeam determines the dynamic characteristics of the machine in X or Y axis, which directly affects the machining efficiency and quality of the machine. The most typical expression of dynamic characteristics is acceleration, and the acceleration is inversely proportional to the mass of the moving parts, so reducing the mass of the moving parts while ensuring the stiffness is an important part of optimizing the machine structure. To address the problem that the cross beam of HIMT50 vertical machining center has structural redundancy, which leads to bad dynamic characteristics, the cross beam is firstly modeled in 3D, and its static and dynamic characteristics are analyzed by Abaqus software, and the data are applied to the optimization module to optimize the structure by topology optimization method. The optimization results show that the deformation and low-order amplitude of the crossbeam are improved with the reduction of its mass, which provides a theoretical basis and reference value for optimizing the crossbeam structure, reducing the overall manufacturing cost and improving the machining efficiency of the machine tool.

Keywords: Vertical machining center finite element analysis, Beam, Static and dynamic characteristics, Topology optimization.

1. Introduction

With the rapid development of aerospace, shipbuilding, engineering machinery, mold and other industries and the requirements of the times of Made in China 2025 and Industry 5.0, the demand for high-end CNC machine tools is increasing day by day[1]. The complex and diversified shapes of various parts have also promoted the development of modern CNC machine tools in the direction of high precision, high speed, high stability and high flexibility[2]. In the machining process of the machine tool, the beam is an extremely important moving part in the whole machine, and its static and dynamic characteristics will have an important impact on the overall processing efficiency and processing accuracy of the machine tool. At present, most machine tool manufacturing enterprises adopt empirical and structural analogy methods when optimizing the design of beam structure, ignoring the importance of lightweight to the structure, which often leads to excessive quality of the designed beam. On the basis of ensuring that the static and dynamic characteristics of the beam are not reduced, the beam structure is optimized, which can improve the dynamic characteristics of the moving parts of the machine tool while reducing the use of materials, and realize the concept of modern industrial green manufacturing.

For the optimization of beam structure, experts and scholars at home and abroad have carried out a lot of research. Guo Linna[3] et al. used the finite element method to analyze the static dynamic of the beam of the moving beam gantry machining center, changed the shape and structure of the internal stiffener of the beam, reduced the deformation of the beam, and increased the first-order natural frequency. On the basis of not changing the original structure of the beam, Bao Li[4]and others proposed three optimization schemes to improve the static and dynamic performance of the whole machine. Zhou Zhencai[5]et al. adopted HyperWorks software to optimize the topology of the beam based on the variable density method with the goal of minimizing structural compliance, which improved the static and dynamic performance of the beam and reduced its weight. Chen Min[6]and others applied topology optimization methods to optimize the internal structure of the beam, and effectively improved the bending stiffness and material utilization rate of the beam by optimizing the position and layout of the stiffeners.

This paper takes HIMT 50 vertical machining center as the object, HIMT 50 is a new type of structural vertical machining center, that is, different from the C-shaped layout of common vertical machining centers, but adopts a viaduct gantry layout, using beams as the carrier of X and Y axes, so that the moving parts are full stroke inclusive, and there is no overturning moment. At the same time, it is also different from the layout of common gantry machining centers, that is, the cross beam adopts the three-point support method of intermediate guide rail + left and right guide rails, instead of the left and right guide rail support layout of common gantry machining centers, so that the stability is greatly improved. However, although the stability of this new structure beam layout is superior, there are also problems of increasing the cross-section and weight caused by the addition of intermediate guide rails, which has a negative impact on the overall dynamic characteristics of the machine tool.

In view of the above problems, the maximum deformation position of the beam under actual working conditions was first found through finite element analysis, and on the basis of ensuring that the static and dynamic performance of the beam was not reduced, the topology of the beam structure and internal rib plate area was optimized, and the static dynamic characteristics were comprehensively analyzed, and the optimal cross-section, wall thickness and reinforcement method were determined to achieve the expected results.

2. Finite Element Analysis of Beams

2.1. Beam material properties

The material used in the HIMT50 vertical machining center beam is gray cast iron HT300, and its material properties are shown in Table 1.

Table 1. HT300 material properties

material	Elastic modulus /GPa	density /(g/cm^3)	Poisson's ratio
HT300	143	7.3	0.27

2.2. Finite element model establishment

2.2.1. Simplify the model

SolidWorks software was used to build a 3D model of the beam component of the HIMT50 vertical machining center. In order to ensure the calculation rate and quality of the subsequent finite element analysis software, it is necessary to remove some component structures and subtle features that have little impact on the final result on the basis of the original model, such as small rounds, threaded holes, shallow grooves, and bosses with little height. The final simplified beam model is shown in Fig.1.

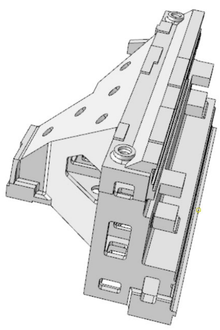


Figure 1. Simplified 3D model of the beam

2.2.2. Meshing

Mesh quality has a certain influence on the accuracy of finite element analysis. To improve analysis accuracy, you need to precisely control the size of the mesh. In general, the finer the meshing, the closer the result is to reality. However, when the grid is refined to a certain size, the accuracy brought by the smaller grid is not significantly improved, and the calculation time of the software will be greatly increased. Therefore, in the meshing stage, it is only necessary to refine the mesh at the key parts of the part.

The meshing adopts a free division method, using quadratic tetrahedral (C3D10) elements, a total of 283216 elements. The meshed finite element model is shown in Fig.2.

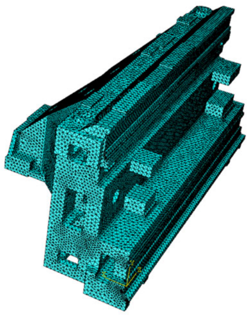


Figure 2. Meshing model of beam

2.2.3. Constraints and load application

When the constraint is applied, considering that the beam is bolted and mounted on the slide, and the slider drives the beam to move horizontally in the X-axis direction of the bed body, the fixed constraint is applied at the joint surface of the beam and the slide. When the load is applied, the effect of the cross slide and components installed on the beam on the beam is equivalent to the load applied to the force on the beam guide; At the same time, taking into account the influence of the self-weight of the beam, when applying the load, a gravitational acceleration of 1g should also be applied in the vertical direction[7].The specific load values are shown in Table 2.

Table 2. Stress on beams

Load type	The numeric size /N
Beam assembly weight	8504.17
Crossbar assembly weight	3241.29
Slide assembly weight	2454.59
X-direction feed force	523.85
Y-direction feed force	555.25
Z-direction feed force	2462.31

2.3. Static characterization

Static stiffness can be used as a basic parameter index to judge the advantages and disadvantages of the structure[8]. The deformation of the cross beam of this machine tool is mainly caused by gravity and cutting force, and the maximum deformation of the cross beam can be obtained at the most dangerous position under the working conditions. Suppose the beam is in the most dangerous condition, that is, when the cross slide is in the middle of the beam. The beam displacement cloud and stress cloud plots solved by finite element analysis are shown in Fig. 3 and Fig. 4, respectively.

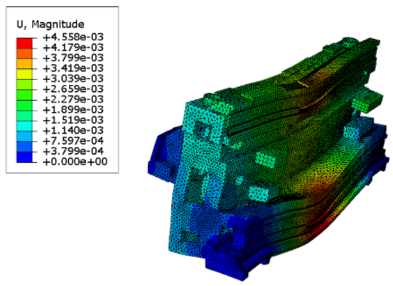


Figure 3. Comprehensive displacement cloud of beams

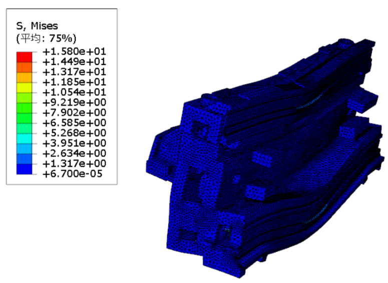
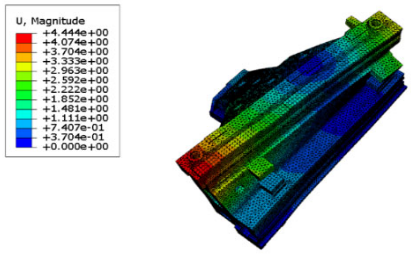


Figure 4. Comprehensive stress cloud of beams

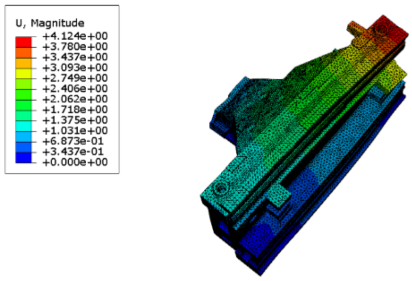
From the static analysis results, it can be seen that under the action of external load, the maximum deformation part of the beam is the middle of the beam, and the maximum deformation is "4.558μm"; The maximum stress of the beam is 15.8 MPa, which is much lower than the allowable stress of 335 MPa of HT300. Except for the maximum deformation part of the beam and the small position of the stress concentration, the deformation and stress of the beam of the vertical machining center are relatively small, indicating that there is a certain optimization space for other regional structures of the beam.

2.4. Dynamic characterization

The dynamic characteristics of the five-axis vertical machining center beam reflect the stability of the structure during external excitation, which has a great influence on the machining accuracy of the machine tool. The basis of machine dynamics analysis is modal analysis, which is mainly used to determine the modal parameters of the machine structure, that is, the natural frequency, damping ratio and modal mode shape during the working process of the machine tool[9]. It can reflect the dynamic deformation of mechanical structure in the working process, and has important guiding significance for the dynamic characteristic design and topology optimization design of the mechanism. Therefore, it is necessary to conduct modal analysis on the beam, and in the actual working conditions, the performance of the beam is mainly affected by the low-order mode. By using the Lanczos method in the Abaqus software to analyze the modal state of the beam, the first two modal mode shapes of the structure are shown in Fig. 5, and the first four frequency values and mode shape descriptions are shown in Table 3.



First-order mode shape



Second mode shape

Figure 5. Fourth-order mode shape diagram of the beam

Table 3. Description of the first four orders of frequency, amplitude and mode shape

Order	Natural frequency /Hz	amplitude /mm	Mode shape description
1	312.67	4.124	The upper part of the beam rotates around the Z axis
2	395.12	4.444	The upper part of the beam is torsionally moving around the Y-axis
3	486.70	3.666	The beam rotates around the X axis
4	527.73	2.069	The beam moves as a whole along the Y axis

3. Optimization of Beam Structure

3.1. Topology optimization methods

Topological optimization is a research method that uses mathematical means to determine the theoretical optimal allocation path of object materials in space[10]. Because it can improve the structural properties while reducing the quality of the structure, the material utilization rate is high, so it is widely used in lightweight research.

Considering that the static stiffness and natural frequency of the structure are contradictory objective functions, it is difficult to make the objective function of stiffness and frequency meet the optimal requirements at the same time when the weight distributions of the two are too different[11]. Combined with the casting difficulty and cost of the beam structure, the optimized structure cannot be completely consistent with the irregular shape of the topology optimization results, so the topology optimization method based on the solid isotropic material with penalization is selected without using the topology optimization method of the evaluation function, and the topology optimization of the beam structure is carried out with the goal of maximizing the static stiffness of the beam and maximizing the natural frequency.

The simp method is an interpolation function constructed between the relative element density and the elastic modulus of the element, and the idea is to use "pseudo-density" as the design variable of the structural finite element, and its value changes continuously between 0~1. By iteratively operating the correlation between the elastic modulus of the pseudo-density element and the objective function, the elements smaller than the set threshold are excluded, and the elements larger than the set threshold are retained[12]. To prevent stiffness matrix singularities in the calculation, the expression[13]is used:

$$E_e = E_e(x_e) = E_{min} + x_e^p(E_0 - E_{min}) \quad x_e \in [0,1] \quad (1)$$

where x_e is the relative unit density; p is the penalty factor; E_0 is the modulus of elasticity of the element; E_{min} is the elastic modulus of the pore material.

3.2. Static stiffness is optimized for the target

In order to ensure that the weight of the beam structure is reduced and the structure obtains better stiffness, the structure of the beam is optimized by maximizing the static stiffness of the beam, and the structure of the beam is optimized by the structural volume percentage as the constraint. When setting the penalty factor, the penalty effect of different penalty factors on the elastic modulus of the material has certain differences, and the penalty effect is shown in Fig. 6. The beam material is HT300 and the elastic modulus E=1.43×10^11 Pa, so the minimum density is set to 0.001 and the penalty factor is 3 in the Abaqus optimization module, which can avoid the phenomenon of poor iterative convergence due to excessive penalty factor[14]. Since the beam needs to be assembled with other components in the overall structure of Lijia, when setting the optimization area, the parts that mate with other components are set as the non-optimization area, and the rest of the parts are the optimization area. Set the minimum flexibility as the objective function, and after many calculations, it is found that when the volume fraction of the beam is 70% of the original, a clearer force transfer path can be obtained. The topology optimization results targeting minimum flexibility are shown in Fig. 7.

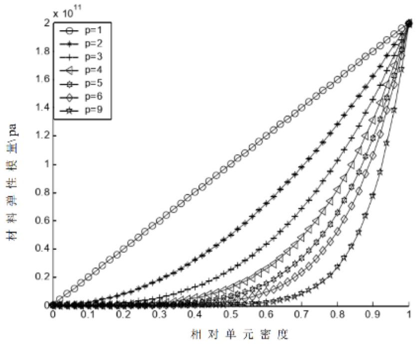


Figure 6. The penalty effect of different penalty factors on elastic modulus

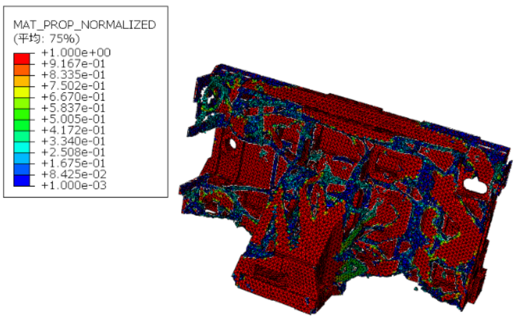


Figure 7. Flexibility minimization is the target optimization result

It can be seen from Figure 7 that the redundancy of the structural materials of the beam is mainly concentrated in the rear surface part of the beam and the small part of the internal rib plate position, so it is preliminarily proposed to modify the structure of these areas and remove the material in the subsequent structural improvement process.

3.3. Natural frequencies are optimized for the target

When the natural frequency of the beam is set as the optimization target, the data obtained in the modal analysis is led to a new topology optimization analysis step, the objective function is set to maximize the natural frequency of the first four orders, and the rest of the settings remain unchanged from the topology optimization parameters aimed at maximizing static stiffness, and the topology optimization results are shown in Fig. 8.

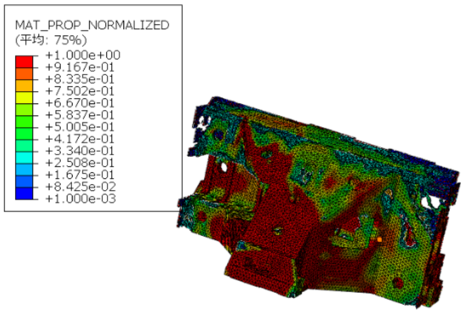


Figure 8. Natural frequency is the target optimization result

It can be seen from Figure 8 that the optimization results aimed at maximizing natural frequencies are significantly different from those obtained by optimizing static stiffness, and most of the less dense positions occur in the middle and top of the beam. According to the optimization results, the middle and upper parts of the beam are preliminarily proposed to modify the structure.

4. Improvement and Analysis of Beam Structure

4.1. Optimized rear beam structure

As mentioned earlier, static stiffness and natural frequency are contradictory objective functions, so it is not possible to simply improve the original structure of the beam with one of the optimization results, and it is necessary to consider the results of the two topological optimizations at the same time. Therefore, for the area where the material removal position is roughly the same in the two optimization results, the material in this area can be directly removed, and for the areas with small material removal rates in the two optimization results, make minor adjustments or no adjustments, which can basically meet the requirements of the two optimization goals, and at the same time, it can not bring in new singular structures to meet the actual machinability and castability. In the process of optimizing the structure, the assembly relationship between the beam and other components must also be considered, and the topology optimization results cannot be fully adopted.

Therefore, under comprehensive consideration, the improvement area of the beam structure is mainly concentrated on the rear upper surface of the beam and part of the internal rib plate position. The improved structure of the beam is shown in Fig. 9.

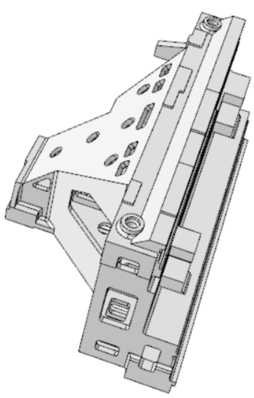


Figure 9. Improved beam structure

4.2. Optimized finite element analysis

In order to ensure the effectiveness of the structural modification, it is necessary to perform static and modal analysis of the improved structure again. During the analysis, parameters such as load application, boundary condition setting, and meshing must be consistent with the previous analysis to increase the reliability of the comparison. The comparison of performance parameters before and after the improvement of the beam structure is obtained, as shown in Table 4.

Table 4. Comparison of performance parameters before and after improvement of beam structure

parameter	Before improvement	After improvement	The degree of change
Maximum deformation /μm	4.558	4.377	-3.97%
weight /Kg	596	575	-3.52%
First-order natural frequency /Hz	312.67	305.02	-2.45%
First-order amplitude /mm	4.124	3.786	-8.20%
Second-order natural frequency /Hz	395.12	377.62	-4.43%
Second-order amplitude /mm	4.444	4.276	-3.78%
Third-order natural frequency /Hz	486.70	485.08	-0.33%
Third-order amplitude /mm	3.666	3.576	-2.45%
Fourth-order natural frequency /Hz	527.73	524.12	-0.68%
Fourth-order amplitude /mm	2.069	2.121	+2.51%

It can be seen from Table 4 that under the condition that the mass of the beam is reduced by 3.52%, the maximum deformation of the beam is reduced by 3.97%, the natural frequency of the first and second orders is slightly reduced, and the natural frequency of the third and fourth orders is almost unchanged, but its low-order amplitude is reduced compared with the original structure, and the optimization effect is more obvious, which meets the optimization purpose of this time.

5. Summary

In order to solve the structural redundancy problem of the vertical machining center beam, the structural quality is reduced without reducing the static and dynamic performance. In this paper, the topology optimization algorithm based on SIMP is adopted to optimize and improve the Lijia beam, and the following conclusions are obtained:

(1) Based on static and modal analysis, the maximum deformation position and structural redundancy area in the original structure are clarified, which provides an optimization direction for subsequent work.

(2) Based on the SIMP variable density method, the topology of the beam was optimized with the goal of maximizing static stiffness and maximizing natural frequency, and the improved beam structure was obtained by comprehensively considering the two optimization results, the assembly relationship of the beam and the machining castability.

(3) The same finite element analysis was performed on the improved structure, and it was found that the mass of the beam decreased, the maximum shape variable decreased, and the low-order amplitude decreased.

Therefore, this method has certain reference value for the optimization design of mesh components similar to beam ribs.

Acknowledgment

Funded by Sichuan Provincial Economic and Information Commission (Sichuan Caijian [2021] No. 106); Key R&D Project in High-tech Field of Sichuan Province (2021YFG0050).

References

[1] ZHANG Z Q,CAO H T. Development of intelligent equipment manufacturing in Liaoning Province under background of “Made in China 2025”[J]. Journal of Shenyang University (Social Science), 2018,20(2):129-132.

[2] YE Y S. Nc Machine tool fault diagnosis methods and their development trend analysis[J].New Curriculum, 2014,(12):219.

[3] GUO L N,ZHENG T C,JU J Q,et al. Structural design and optimization in the beam of moving beamlongmen processing center[J].Modern Manufacturing Engineering, 2017,(6):90-93.

[4] BAO L,LI X C,TONG J. Improvement design of weak link of gantry five axis machining center[J]. Machine Tool & Hydraulics, 2015, 43(11):34-37.

[5] ZHOU Z C,LI T,WANG T L,et al. Topology optimization design of machine tool beam based on HyperWorks[J]. Mechanical & Electrical Engineering Technology, 2018, 47(5):9-12,114.

[6] CHEN M,ZHANG H W,CHEN Q Y,et al. Finite element analysis and optimized design of the gantry machining center crossbeam [J]. Mechanical & Electrical Engineering Technology, 2018, 47(4):40-41,119.

[7] GAO Z L,QIU Z X,REN D,et al. Structure design and optimization for crossbeam of bridge gantry milling mac-hine[J].Chinese Journal of Engineering Design, 2019, 26 (1): 56-64.

[8] ZHANG G H,HE Y,FEI S W,et al. Static and dynamic ch-aracteristics analysis and optimization forcrossbeam of CNC gantry milling machine[J].Modern Manufacturing Engineering,2021,(3):146-151.

[9] DAI G L. Therma l performance analysis of NC machine tool structure and topology optimization for supporting c-omponents[J].Mechanical Research & Application, 2013, 26 (5): 131-134.

[10] ZHANG H,XU Y,LIAO Y H,et al. Static and dynamic ch-aracteristics analysis and Rib plate topology optimization of Five-axis vertical machining c-enter[J].Modular Mach-ine Tool & Automatic Manufacturing Technique,2021, (9):33-37.

[11] Costa G, Montemurro M. Eigen-frequencies and harmon-ic responses in topology optimization: A CAD-compatibl-e algorithm[J].Engineering Structures, 2020, 214:110602.

[12] YANG F,HU Q,WANG H,et al. Optimization design for worktable stiffener of heavy-duty vertical lathe[J].Machi-ne Tool & Hydraulics, ,2022,50(1):107-111.

[13] Liu K, Tovar A. An efficient 3D topology optimizat-ion code written in Matlab[J].Structural and Multidis-ciplinary Optimization,2014,50(6):1175-1196.

[14] Srinivas G L, Javed A. Topology optimization of in-dustrial manipulator-link considering dynamic loading[J].Materials Today:Proceedings,2019,18:3717-3725.