中國機械工程學刊第四十六卷第一期第 51~61 頁(民國一百一十四年)
Journal of the Chinese Society of Mechanical Engineers, Vol.46, No.1, pp51~61 (2025)

Comprehensive Modal Analysis, Experimental Validation, and Topology Optimization of High-Precision Surface Grinding Machine

Yi-Cheng Ye*, Shinn-Liang Chang**, Tzu-Chi Chan***and Keng-Chang Chang****

Keywords: experimental modal analysis, natural frequency, surface grinding machine, topology optimization.

ABSTRACT

This study focuses on the dynamic characteristics of a high-precision surface grinding machine. Modal analysis using Finite Element Analysis (FEA) was employed to determine the fundamental frequencies of the structure. An Experimental Modal Analysis (EMA) was conducted to validate the vibration characteristics and prevent resonance. Modal analysis confirms primary and secondary natural frequencies of 68.2 and 134.3 Hz, respectively, with a discrepancy of <10% compared to experimental results. The topology optimization of the critical components increased the primary and secondary natural frequencies by 3.7% and 5.3%, respectively, raising the corresponding values to 70.8 and 141.8 Hz. This optimization effectively reduced the risk of resonance in the 60–70 Hz range.

INTRODUCTION

Surface grinding machines, which employ abrasive wheels for material removal, exhibit exceptional precision and capability to achieve ultralow surface roughness. They constitute the critical final stage of the manufacturing processes, particularly for components that require the highest precision. In the field of machine tool design and production, structural rigidity and vibration control remain focal points for rigorous examination. These determinants directly influence the machining precision and have profound implications for machine longevity and maintenance costs. To address these critical considerations, the proactive integration of Finite Element Analysis (FEA) emerged as a pivotal design methodology. This enables the accurate anticipation of the dynamic attributes of the structure of the machine and, in turn, facilitates the precision-targeted reinforcement of its structural integrity. The outcome was a noticeable enhancement in both production efficiency and product quality, conferring considerable value in engineering.

In the field of grinding machines, vibration analysis and structural improvement are extensively explored. Myers et al. (2003) investigated the key role of structural rigidity of machine tools in precision machining. They reported that superior static rigidity ensured accurate dimensions and shapes during machining, whereas exceptional dynamic rigidity improved surface quality and machine protection. Advanced Finite Element Analysis (FEA) was employed to assess and confirm structural rigidity, with a focus on milling machines, thus validating its practical effectiveness. Aurich et al. (2009) focused on the interaction of grinding processes with machine structures by studying vibrations, deflections, and thermal effects. Using various coupling methods, they enhanced simulations to improve the design of processes and machinery, ensuring better workpiece quality and stability. Future research should aim for a deeper understanding and optimization of grinding processes, including thermal parameter exchange and cooling lubricant effects, in coupling processes and machine models. Pakzad et al. (2012) investigated the dynamic characteristics and natural frequencies of the FGMs. They analyzed the modal properties of the machine using CATIA and ANSYS. Modal testing of the structure was conducted, and the results were compared with those of modal analysis, highlighting their consistency. Chen et al. (2017) employed a method that combined experimental modal and finite element

Paper Received November, 2023. Revised May, 2024. Accepted July, 2024. Author for Correspondence:Tzu-Chi Chan.

** Graduate student, Department of Power Mechanical Engineering, National Formosa University, Yunlin, Taiwan 632301, ROC.*

***Professor,Department of Power Mechanical Engineering, National Formosa University, Yunlin, Taiwan 632301, ROC.*

****Associate Professor, Department of Mechanical and Computer-aided Engineering, National Formosa University, Yunlin Taiwan ,632301, ROC.*

***** President, EQUIPTOP, Taichung, Taiwan, 421016, ROC.*

analyses to assess a grinding machine. The experimental modal analysis revealed the key frequency-related parameters, whereas the finite element analysis provided information on the natural frequencies and stiffness. Comparing the theoretical and experimental models through multiple iterations reduced costs and identified design weaknesses for improvement. Muhammad et al. (2020) optimized a diesel engine connecting rod for weight reduction under a 100 N static load. They compared the deformations, stress, strain, and safety factors before and after achieving 60% weight reduction. Zheng et al. (2020) employed the Finite Element Method (FEM) to create a finite element model for a small grinding fixture. They calculated the first five natural frequencies and damping ratios of a small grinding fixture and verified the accuracy of the identification results using modal judgment criteria. This process mitigates the resonance issues during machining. Chan et al. (2022) optimized the performance of the precision grinding machine. Modal analysis revealed modal frequencies of 26.9 and 27.7 Hz. By increasing the saddle height by 200 mm, the modal frequency of the machine improved by 26% to 35.0 Hz, avoiding resonance below 30 Hz.

This study analyzed and experimented using a high-precision flat-grinding machine produced by EQUIPTOP HITECH Corp. In the experiments, an impact hammer served as the external exciter and three-axis accelerometers were employed to measure the vibration signals of the machine. Subsequently, the simulation and experimental results were validated and compared to determine the machine rigidity. Areas with relative structural weaknesses were identified for topology optimization to achieve weight reduction, improved rigidity, and natural frequency enhancement for ensuring that the product meets the precision requirements, thus upholding rigorous quality standards in precision technology.

MATERIALS AND RESEARCH METHODS

3D Modeling and Material Selection.

This study involved the three-dimensional modeling of a high-precision surface grinding machine manufactured by EQUIPTOP HITECH CORP using SolidWorks, as depicted in Figure 1. Subsequently, the entire system underwent modal analysis using ANSYS analysis software to mitigate the potential for resonance. Cast iron and steel materials were employed for subsequent simulation analyses, with their respective properties detailed in Table 1. Additionally, the specifications for each axis drive of the grinding machine were meticulously detailed in Table 2.

Finite Element Balance Equations

The finite-element method (FEM) is a numerical analysis technique used to solve various physical problems in engineering and science. The core concept of the finite element method involves subdividing a complex, continuous domain into numerous small finite elements, which can be adjusted based on the nature of the problem. Subsequently, the behavior of these finite elements is transformed into an algebraic equation system. A schematic of the FEM is shown in Figure 2.

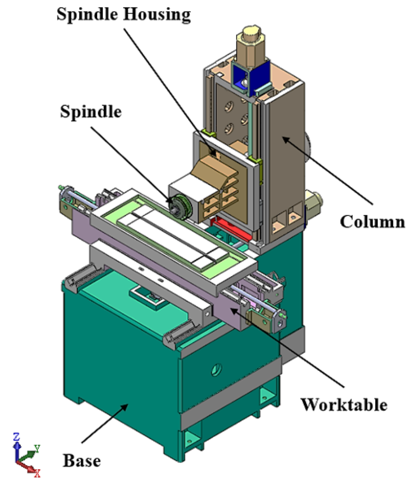


Fig. 1. 3D model of the high-precision surface grinding machine.

Table 1. Material properties.

Properties	Cast iron	Steel
Density (kg/m^3)	7150	7850
Young's modulus (GPa)	92.4	205
Poisson's Ratio	0.21	0.29

Table 2. Equipment specifications.

Equipment model	ESG-818CNC
X-axis travel speed	0~20 m/min
X-axis maximum travel range	510 mm
Y-axis travel speed	0~3 m/min
Y-axis maximum travel range	260 mm
Z-axis travel speed	0~3 m/min
Z-axis maximum travel range	390 mm
Spindle rotation speed	3600 rpm
Spindle motor power	5 HP

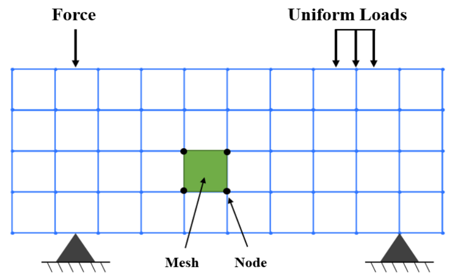


Fig. 2. Schematic of the finite element analysis.

In a static structural analysis, each element must satisfy equilibrium conditions. According to the principles of mechanics, for equilibrium of an individual element, the sum of the internal forces within the element must be zero when considering the applied external forces. Therefore, to describe the internal force equilibrium within an element, the equilibrium equation can be expressed as

$$[K]\times\{U\}=\{F_t\}, \tag{1}$$

$$\{F_t\}=\{F_f\}+\{F_r\}, \tag{2}$$

where [K] represents the stiffness matrix, {Ft} is the element nodal external force vector, {U} is the element nodal displacement vector, {Fr} is the reaction load vector, and {Ff} is the external force vector.

Vibration System Model

The dynamic analysis of complex structural systems can be effectively generalized using a single-degree-of-freedom system. This simplification aids in a better prediction of the structural response behavior. A single-degree-of-freedom system typically consists of a mass, spring elements storing potential energy, and a damper, as shown in Figure 3. The equation for a single-DOF system can be expressed as

$$[M](\ddot{x})+[C](\dot{x})+[K](x)=\{F(t)\}, \tag{3}$$

where [M] represents the System Mass Matrix, [C] is the Structural Damping Matrix, [K] stands for the Stiffness Matrix, (x) is the Nodal Displacement Vector, $(\dot{x})$ is the Nodal Velocity Vector, and $(\ddot{x})$ is the Nodal Acceleration Vector.

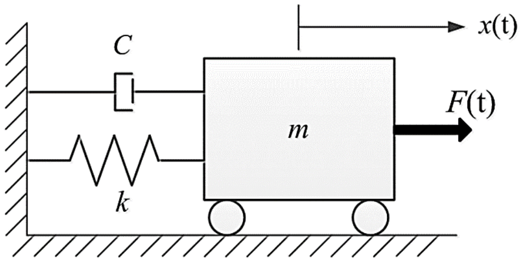


Fig. 3. Single degree of freedom vibration system model.

In this study, owing to the intricate nature of the analytical model, damping characteristics were incorporated into the material properties of the system to accurately simulate their dynamic response. The specific damping ratio settings are shown in Table 3.

Table 3. Material damping ratio setting.

	Cast iron	Steel
Damping Ratio	0.003	0.0005

Contact Configuration and Meshing Setting

In this study, the ANSYS Workbench modal solver was employed for modal analysis to identify vibration modes and predict natural frequencies, thereby facilitating structural design optimization. Initially, contact settings were applied to the surface grinding machine. The Contacts feature within Ansys Workbench was utilized for this purpose, with the completed model imported from Solidworks, and the Bonded contact type selected to simulate contact. An illustration of the overall contact configuration is provided in Figure 4.

Subsequently, meshing was performed for the surface grinding machine. Tetrahedral and Quadratic methods were utilized for meshing configuration. Mesh sizes were set as follows: 30mm for the base and column, 15mm for the worktable and spindle housing, 10mm for the drive module, and 5mm for other components. The overall mesh division is shown in Figure 5. With a total node count of 1,226,618 and an element count of 719,268, such mesh settings ensure a balance between computational efficiency and accuracy, facilitating comprehensive finite element analysis of the entire machine.

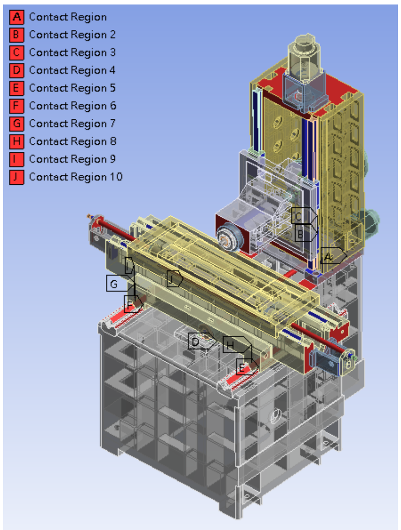


Fig. 4. Schematic of contact configuration.

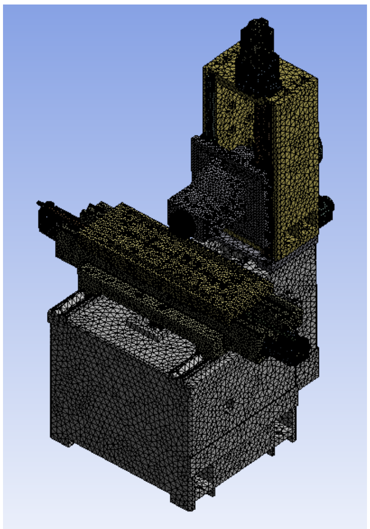


Fig. 5. Schematic of overall mesh division.

Modal Analysis

Based on the preparatory work involving contact settings and meshing for the machine, modal analysis was conducted on the surface grinding machine to ascertain vibration modes and natural frequencies, further facilitating structural design optimization. The machine base was initially constrained as a boundary condition, and external loads were excluded, as shown in Figure 6. The analysis was conducted within a frequency range of 1–200 Hz. The results revealed that the primary mode, Mode 1, occurs at 68.2 Hz, characterized by the vertical column and spindle oscillating along the X-axis, as shown in Figure 7. The secondary mode, Mode 2, was observed at 134.3 Hz, featuring the vertical column and spindle twisting around the Z-axis, with the base and working platform swaying along the X-axis, as shown in Figure 8. Given that the spindle of the machine operates at a fixed speed of 3600 rpm, equivalent to a natural frequency of 60 Hz, avoiding operating frequencies near this resonance range is crucial to enhance the machining precision.

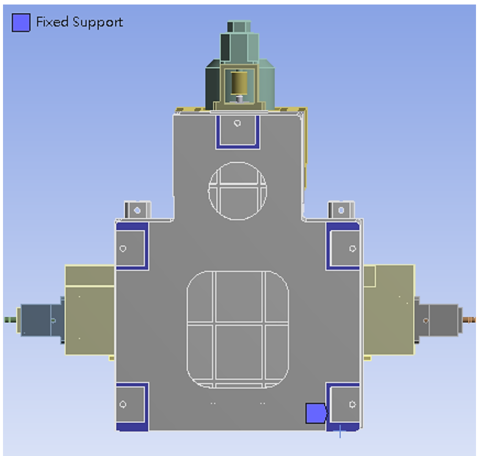


Fig. 6. Modal analysis boundary condition settings.

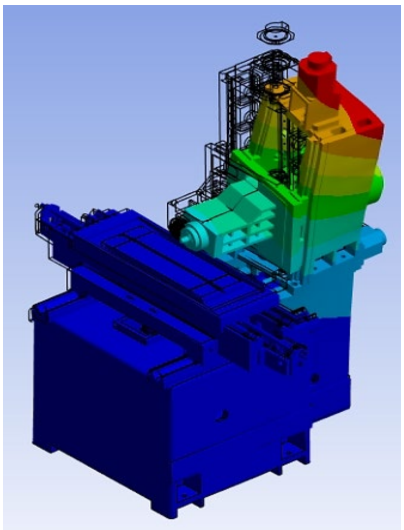


Fig. 7. Mode 1: 68.2 Hz.

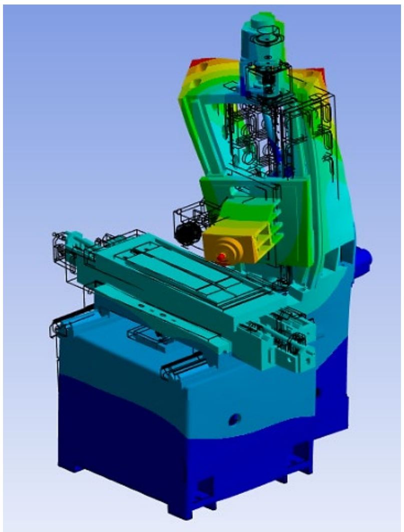


Fig. 8. Mode 2: 134.3 Hz.

Experimental Modal Analysis

To obtain the dynamic characteristics of the structure of the high-precision flat grinding machine, this study employed the "Experimental Modal Analysis," also known as the "Modal Impact Testing" method. In this experiment, an impact hammer was used to strike the spindle seat in the x-axis direction following the principle of external excitation, which induces structural vibration. Three-axis accelerometers were used to measure the vibration signals at various locations and collect structural vibration characteristics. Modal impact testing specifications are shown in Table 4. Modal impact testing of the structure provides a clear and intuitive understanding of its dynamic properties and is a reference for subsequent topological optimization designs. A diagram of the modal test is shown in Figure 9.

Table 4. Modal impact testing specifications.

Impact hammer	
Operating temperature range (°C)	-20~70
Force measurement range (N)	20000
Sensitivity at 100 Hz (mV/N)	0.2
Resonance frequency (kHz)	20
Stiffness (kN/μm)	2.7
Three-axis accelerometer	
Output type	IEPE
Maximum measurement range (g) (m/s^2)	±50
Maximum frequency (Hz) (±5%)	6000
Sensitivity of X-axis (mV/g)	103.9
Sensitivity of Y-axis (mV/g)	104.0
Sensitivity of Z-axis (mV/g)	104.6
Analysis software	ME'scope

Fig. 9. Schematic for modal testing.

During modal testing, careful consideration of accelerometer placement is crucial to ensure coverage of primary vibration modes and comprehensive measurement of structural vibration characteristics. Therefore, in this study, priority was given to selecting areas with larger primary amplitudes, as these regions may have a greater impact on the overall vibration characteristics of the structure. Additionally, it is necessary to ensure that accelerometers are evenly distributed across the main structure to ensure comprehensive measurement of the vibration characteristics of the entire structure without overlooking any critical locations. Through such placement strategy, we can effectively verify the accuracy of modal analysis and obtain more comprehensive structural vibration characteristic data. The layout is shown in Figure 10.

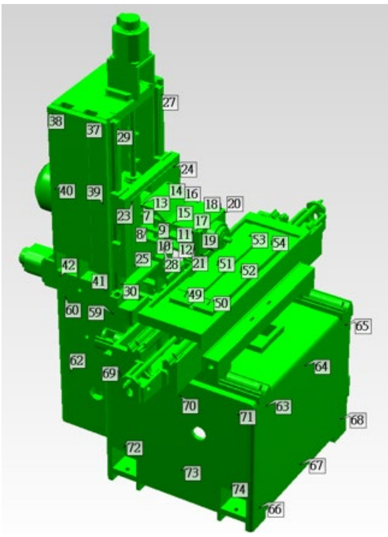


Fig. 10. Modal testing point layout diagram.

As shown in Figure 11, the vibration mode of the experimental findings revealed Mode 1 at 71.2 Hz, indicating that the main shaft and column oscillated along the X-axis. The vibration mode showing the column and main shaft twisting around the z-axis, as shown in Figure 12, was identified as mode 2. Its frequency was 146.0 Hz.

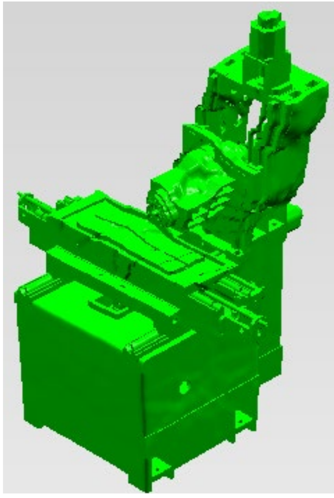


Fig. 11. Modal testing, mode 1:71.2 Hz.

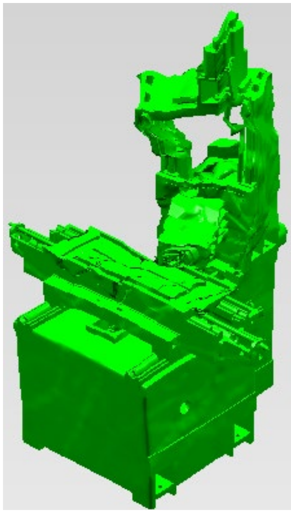


Fig. 12. Modal testing, mode 2:146.0 Hz.

Modal Testing and Modal Analysis Comparison

In this study, a comparative analysis was conducted between the results of the finite element modal analysis and modal testing. Table 5 provides the results of the comparison. The natural frequencies and mode shapes of the first and second modes correspond closely to those of the two methods. Particularly, the first-mode natural frequency obtained from modal analysis was 68.2 Hz, while the experimental modal testing yielded a value of 71.2 Hz, resulting in a percentage error of 4.2%. Similarly, for the second mode, the modal analysis indicated a natural frequency of 134.3 Hz, whereas modal testing measured a frequency of 146.0 Hz, resulting in a percentage error of 8.0%.

Table 5. Comparison of FEA and EMA Results.

Mode	FEA(Hz)	EMA(Hz)	Error(%)
1	68.2	71.2	4.2
2	134.3	146.0	8.0

In summary, the errors in both the simulation analysis and experimental measurements were within 10%. These results indicated that the boundary conditions set in the simulation analysis aligned with those of the actual machine, thus validating the reliability and accuracy of the analytical findings.

TOPOLOGY OPTIMIZATION ANALYSIS

The primary objective of this study in topological optimization is to achieve structural lightweighting while enhancing the structural rigidity and natural frequencies, ensuring the performance and precision of the machine tool. The goal was to increase the first-order natural frequency of the structure above 70 Hz, thereby avoiding resonance frequencies during machine operation. Furthermore, topological optimization and design analysis were conducted on three critical components: the base, column, and spindle housing. Comparative assessments of the characteristics before and after optimization were also performed.

Due to the outcomes of topological analysis results, which solely provide recommendations for areas to be removed, it is essential to consider the practical installation requirements of the machinery during the design phase. Specifically, between the spindle housing and the column, there are other components that need to be secured, making direct removal of the suggested areas unfeasible. Therefore, to meet the practical engineering installation demands and ensure structural safety and stability, the design not only incorporated the recommendations derived from the topological analysis for area removal but also considered the incorporation of reinforcement ribs. This comprehensive approach aims to guarantee the overall structural stability and rigidity.

Topology Optimization of the Base

The boundary conditions for the topological optimization of the base are set according to the actual machine setup: (1) The footings of the base are fixed to simulate the placement of the machine on the ground, restricting its displacement. (2) A load of 5069 N was applied to the base slide rails to simulate the worktable weight. (3) A load of 4260 N was applied at the rear of the base, where it is interlocked with the column, simulating the weight borne by the upper part, including the column and spindle housing. (4) The overall volume was maintained at 95% of the original volume. Furthermore, based on the results of the topological analysis, removable regions for model redesign were identified, as shown in Figure 13.

After conducting a topological analysis and implementing the recommended modifications to the base, a comparative analysis of the mass and modal characteristics before and after optimization was performed. Based on model features and analysis results, the overall mass of the base decreased from the initial 776.4 to 771.4 kg, resulting in a reduction of 5.0 kg. The modal frequency 1 increased from the original 229.9 to 296.7 Hz, showing a 22.5% improvement. Similarly, modal frequency 2 increased from the initial 237.9 to 322.6 Hz, indicating a 26.3% enhancement. A before-and-after comparison of the base topological optimization results is provided in Table 6.

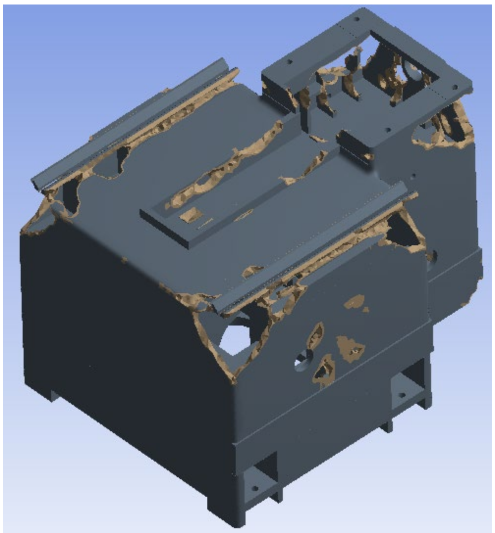
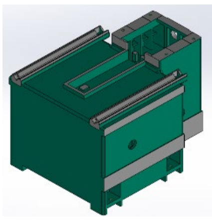
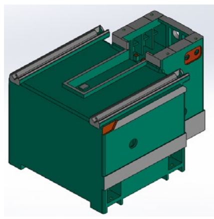
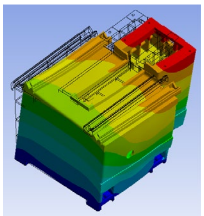
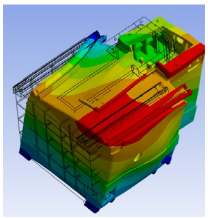
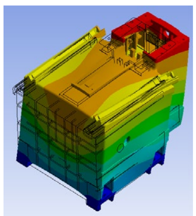
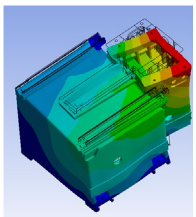


Fig. 13. Base topological analysis suggestions.

Table 6. Comparison before and after base topological optimization.

Original	Topology	Error
		-0.6%
Mass: 776.4 kg	Mass: 771.4 kg	
		+22.5%
Mode 1: 229.9 Hz	Mode 1: 296.7 Hz	
		+13.3%
Mode 2: 237.9 Hz	Mode 2: 322.6 Hz	

Topology Optimization of the Spindle Case

The boundary conditions for the topological optimization of the spindle housing were configured based on the actual machine setup: (1) Fixed the faces interlocked with the z-axis slide and z-axis screw to simulate the actual fixation of the spindle housing. (2) A load of 202 N was applied to the midhole face to simulate the weight of the spindle axis. (3) A load of 475N was applied at the rear face connected to the motor to simulate the weight of the motor fixation. (4) The overall volume was maintained at a minimum of 95% of the original volume. Furthermore, based on the results of the topological analysis, the regions eligible for removal were identified for model redesign, as shown in Figure 14.

After subjecting the spindle to a topological analysis and implementing the recommended modifications, a comparative analysis of the mass and modal characteristics before and after optimization was performed. Based on model features and analysis, the overall mass of the spindle housing decreased from the initial 94.1 to 92.2 kg, resulting in a reduction of 1.9 kg. Modal frequency 1 increased from the original 604.7 to 766.2 Hz, indicating a 21.1% improvement. Similarly, modal frequency 2 increased from the initial 739.1 to 870.3 Hz, indicating a 15.1% enhancement. A before-and-after comparison of the topological optimization results of the spindle housing is provided in Table 7.

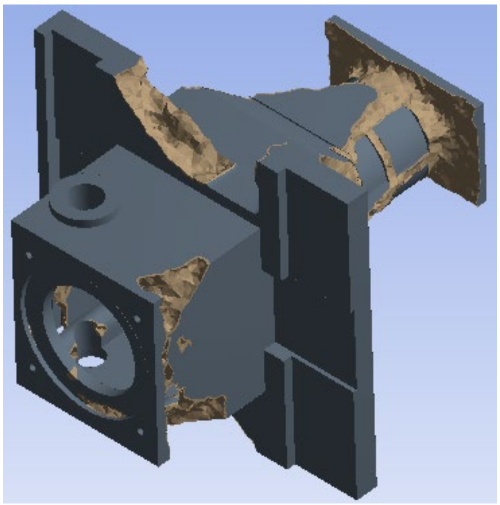
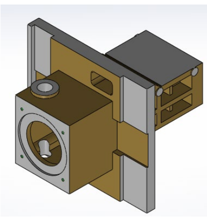
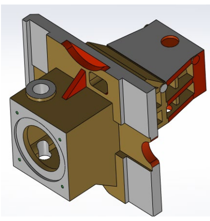
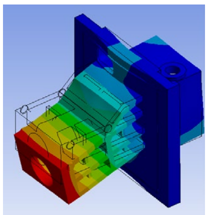
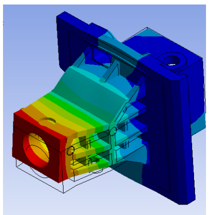
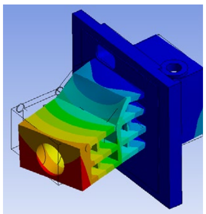
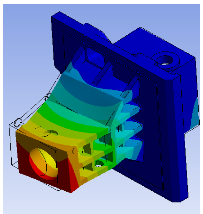


Fig. 14. Spindle case topological analysis suggestions.

Table 7. Comparison before and after spindle case topological optimization.

Original	**Topology**	**Error**
		-2.0%
Mass: 94.1 kg	Mass: 92.2 kg	
		+21.1%
Mode 1: 604.7 Hz	Mode 1: 766.2 Hz	
		+15.1%
Mode 2: 739.1 Hz	Mode 2: 870.3 Hz	

Topology Optimization of the Column

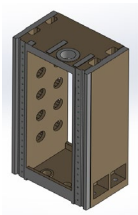
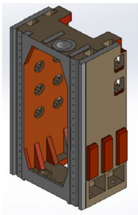
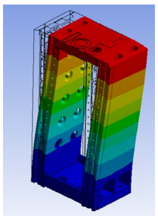
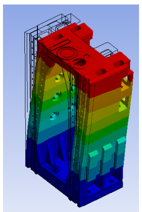
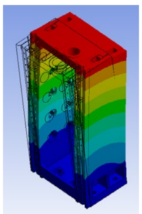
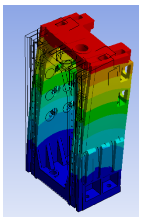
The boundary conditions for the topological optimization of the column were configured based on the actual machine setup: (1) the bottom of the column was fixed to the connection point with the base, the actual locking of the column onto the base was simulated, and its displacement was restricted. (2) A load of 2066 N was applied to the faces where the column was interlocked with the linear rails to simulate the weight of the spindle housing fixation. (3) The overall volume is maintained at a minimum of 95% of the original volume. In addition, based on the results of the topological analysis, the regions eligible for removal were identified for model redesign, as shown in Figure 15.

Fig. 15. Column topological analysis suggestions.

Following the topological analysis of the column and implementation of the recommended modifications, we conducted a comparative analysis of the mass and modal characteristics before and after optimization. Given the critical load-bearing role of the column within the overall structure and its prominent position in the modal analysis results, the optimization design emphasized not only reducing the volume to decrease the weight but also introducing reinforcing ribs to enhance the structural rigidity.

Based on the features of the model and the analysis, it is observed that the overall mass of the column slightly increased from the initial 199.4 to 199.7 kg, resulting in a 0.3 kg increment. Modal frequency 1 increased from the original 75.4 to 91.1 Hz, showing a 17.2% improvement. In addition, modal frequency 2 increased from the initial 153.6 to 170.9 Hz, reflecting a 10.1% enhancement. A before-and-after comparison of the column topological optimization results is shown in Table 8.

Table 8. Comparison before and after column topological optimization.

Original	**Topology**	**Error**
		+0.2%
Mass: 199.4 kg	Mass: 199.7 kg	
		+17.2%
Mode 1: 75.4 Hz	Mode 1: 91.1 Hz	
		+10.1%
Mode 2: 153.6 Hz	Mode 2: 170.9 Hz	

Comparison and Discussion of Overall System Topological Results

After conducting a topological optimization analysis of the key components and implementing design modifications in line with the optimization recommendations, the rigidity of the optimized components significantly improved. Subsequently, the three optimized components were integrated into the original machine structure. A dynamic characteristic analysis was then performed on the postoptimization structure, followed by a comprehensive comparative analysis of the original design. The complete postoptimization system model is shown in Figure 16.

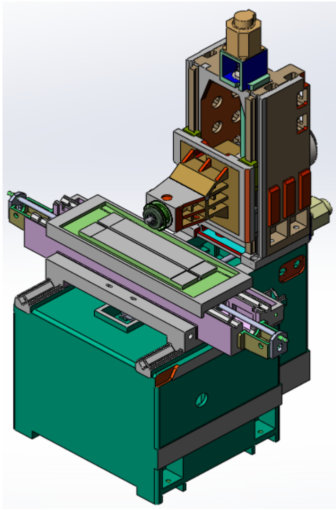


Fig. 16. High-precision surface grinding machine 3D model after topological optimization.

Modal analysis was performed on the postoptimization complete machine structure and compared with the preoptimization results. The study revealed a reduction in the mass of the entire machine structure, decreasing from the original 1552.5 to 1546.3 kg, resulting in a total reduction of 6.2 kg. As the primary focus of this research was to enhance the structural rigidity, rib reinforcements were added to the topological design of the column. Consequently, the mode 1 frequency increased from the original 68.2 to 70.8 Hz, marking a 3.7% improvement, with the corresponding mode shape shown in Figure 17. The mode 2 frequency also increased from its original value of 134.3 to 141.8 Hz, reflecting a 5.3% improvement, with the corresponding mode shape shown in Figure 18.

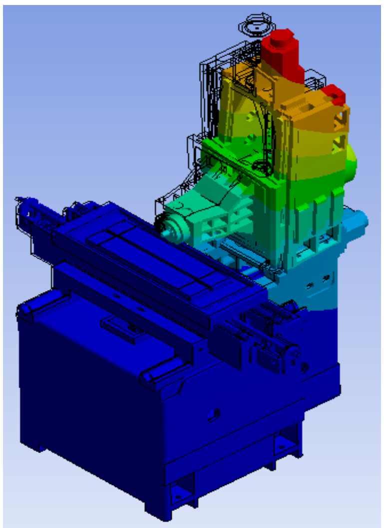


Fig. 17. After topological, mode 1:70.8 Hz.

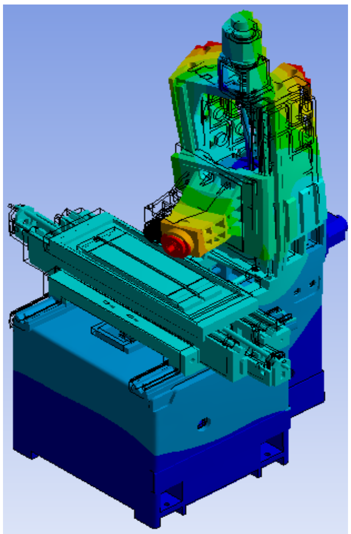


Fig. 18. After topological, mode 2:141.8 Hz.

Next, a static rigidity analysis of high-precision surface grinding machine models was performed before and after topological optimization. The static rigidity analysis in this study was based on the actual machine setup with the following boundary conditions: (1) The base feet of the machine were fixed, simulating the actual placement of the machine on the ground, restricting its displacement. (2) The loading condition considers the self-weight of the machine owing to a gravitational acceleration of 9.81 m/s^2. The boundary conditions are shown in Figure 19.

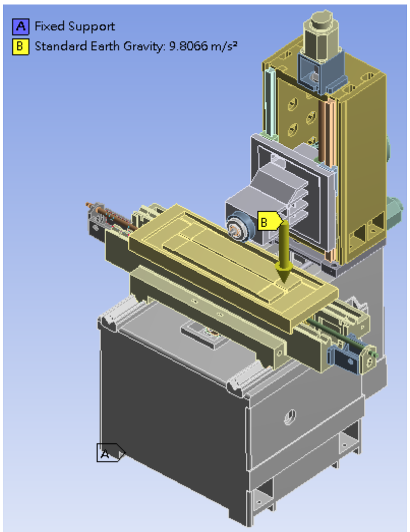


Fig. 19. Boundary condition setup for static rigidity analysis.

From the results of the static rigidity analysis, it is evident that before topological optimization, the entire machine structure exhibited a maximum deflection of 5.8 μm at the spindle end owing to the influence of its own material properties and the effects of gravity, as shown in Figure 20. However, after undergoing topological optimization, the improved structure displayed a maximum self-weight deflection of 5.7 μm at the spindle end, as shown in Figure 21. A comparison of the static deflections before and after topological optimization revealed a reduction of 1.7% in the spindle deflection, as shown in Table 9.

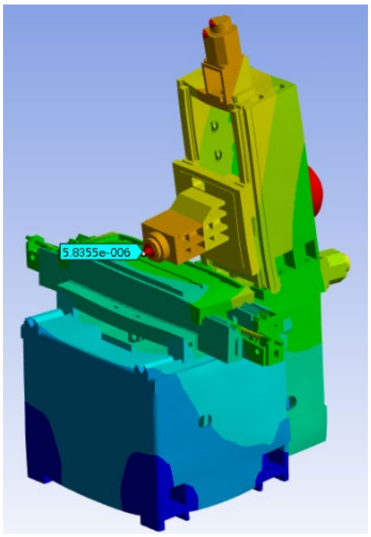


Fig. 20. Static rigidity analysis of the original structure, spindle displacement: 5.8 μm.

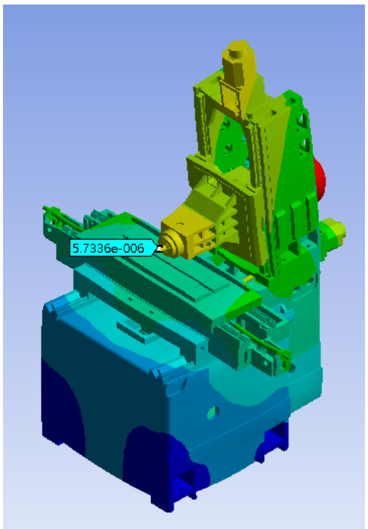


Fig. 21. Static rigidity analysis of the structure after topological spindle displacement: 5.7 μm.

Table 9. Comparison before and after column topological optimization.

	Original	**Topology**	**Error**
Mode 1 (Hz)	68.2	70.8	+3.7%
Mode 2 (Hz)	134.3	141.8	+5.3%
Deformation (um)	5.8	5.7	-1.7%
Mass (kg)	1552.5	1546.3	-0.4%

CONCLUSIONS

To enhance the performance of the high-precision surface grinding machine, finite element analysis and modal impact experiments were conducted in this study to perform static rigidity and modal analyses. In addition, dynamic characteristic tests were performed to validate the accuracy and reliability of the analyses, which served as a reference for the subsequent topological optimization. Modal analysis results revealed that the structural modal frequencies 1 and 2 were 68.2 and 134.3 Hz, respectively. Experimental modal testing showed frequencies of 71.2 and 146.0 Hz for modes 1 and 2, respectively. The relative differences were 4.2% and 8.0%, both within a tolerance of 10%, thus validating the accuracy of the analyses compared to the experimental results.

Moreover, the results of topological optimization indicated improvements in the structure. Particularly, modal frequency 1 increased from 68.2 to 70.8 Hz, showing a 3.7% improvement, while modal frequency 2 increased from 134.3 to 141.8 Hz, showing a 5.3% improvement. Furthermore, the static displacement of the spindle decreased from 5.8 to 5.7 µm, reflecting a 1.7% reduction. The overall structural mass was reduced from 1552.5 to 1546.3 kg, a decrease of 5.0 kg.

Given that the operational speed of the experimental machine in this study was 3600 rpm, which corresponds to 60 Hz, the modal frequency 1 aligned closely with the operating frequency of the machine and was prone to resonance. Through topological optimization analysis and design, Mode 1 shifted away from the resonance zone, achieving an improved structure with reduced weight and increased natural frequencies. The optimization of machine rigidity was instrumental in minimizing machining accuracy losses stemming from resonance phenomena.

REFERENCES

Aurich, J. C., Biermann, D., Blum, H., Brecher, C., Carstensen, C., Denkena, B., Klocke, F., Kroger, M., Steinmann, P., and Weinert, K., "Modelling and Simulation of Process: Machine Interaction in Grinding," *Prod. Eng.*, Vol. 3, pp. 111-120 (2009).

Bendsoe, M. P., & Kikuchi, N., "Generating Optimal Topologies in Structural Design Using a Homogenization Method," *Comput. Methods Appl. Mech. Eng.*, Vol. 71, No. 2, pp. 197-224 (1988).

Chen, D. C., Chen, M. F., Kang, J. H., & Lai, C. C., "Vibration Characteristics and Modal Analysis of a Grinding Machine," *J. Vibroengineering*, Vol. 19, No. 8, pp. 6288-6300 (2017).

Chan, T. C., Chang, K. C., Chang, S. L., Chiang, P. H., "Simulation, Modeling, and Experimental Verification of Moving Column Precision Grinding Machine," *J. Chin. Inst. Eng.*, Vol. 45, No. 1, pp. 54-64 (2022).

Myers, A., Ford, D. G., & Xu, Q., "Finite Element Analysis of the Structural Dynamics of a Vertical Milling Machine," *WIT Trans. Eng. Sci.*, Vol. 44, pp. 431-440 (2003).

Masutage, V. S., & Kavade, M. V. "A Review Vibrating Screen and Vibrating Box: Modal and Harmonic Analysis." *Int. Res. J. Eng. Technol.*, Vol. 5, pp. 401-403 (2018).

Muhammad, A., Ali, M. A., Shanono, I. H., "Design Optimization of a Diesel Connecting Rod," *Materials Today: Proceedings*, Vol. 22, pp. 1600-1609 (2020).

Pakzad, S., Rajab, A. K. S., Mahboubkhah, M., Ettefagh, M. M., & Masoudi, O., "Modal Analysis of the Surface Grinding Machine Structure through FEM and Experimental Test," *Adv. Mater. Res.*, Vol. 566, pp. 353-356 (2012).

Sun, Y., Zhang, Z., Zhang, J., Jin, X., Xu, B., Deng, Y., Zhao, G., "Effects of Transient Slamming and Harmonic Swings on the Marine Compound NC Machine Tool," *Ocean Eng.*, Vol. 108, pp. 606-619 (2015).

Srinivas, G. L., Javed, A., "Topology Optimization of Industrial Manipulator-Link Considering Dynamic Loading," *Materials Today: Proceedings*, Vol. 18, pp. 3717-3725 (2019).

Teimouri, M., Mahbod, & M., Asgari, M., "Topology-Optimized Hybrid Solid-Lattice Structures for Efficient Mechanical Performance," *Structures*, Vol. 29, pp. 549-560, Elsevier (2021).

Zheng, D., Zhang, Q., Zhang, J., Yao, H., & Yin, S., "Finite Element Analysis and Testing on the Vibration Mode of Small Grinding Carriage in High-Efficiency CNC Grinding Machine," *In Journal of Physics: Conference Series*, Vol. 1653, No. 1, p. 012068 (2020).

高精密平面磨床之模態分析、實驗驗證與拓樸最佳化研究

葉奕承
國立虎尾科技大學動力機械工程學系

張信良
國立虎尾科技大學動力機械工程學系

詹子奇
國立虎尾科技大學機械與電腦輔助工程學系

張耿彰
衆程科技股份有限公司

摘 要

本研究旨在探討高精密平面磨床的動態特性。使用有限元素分析（FEA）的模態分析確定結構的自然頻率，再透過模態衝擊實驗（EMA）量測實際振動特性並且加以驗證，以防止機台產生共振現象。模態分析結果顯示前兩階主要自然頻率分別為68.2Hz與134.3Hz，並且分析與實驗的誤差僅低於10%。透過拓樸優化結構的關鍵部件，使整體結構前兩階自然頻率分別提升3.7%與5.3%。並且模態1提升至70.8Hz，模態2提升至141.8Hz。這種拓樸優化設計提高結構的模態頻率，成功避免結構在60Hz~70Hz區間內的共振。