Structural and Modal Analysis of the Unloading Robot on the Broaching Machine

ABSTRACT

In order to meet the actual production of a machining company in the low production efficiency, labor intensity, process flow complexity and other real problems, the broaching machine processing upgraded and designed a broaching machine for the production of automatic loading and unloading device. This paper analyzes the layout scheme and workflow of automatic loading and unloading, designs the overall structure of the loading and unloading device and key components, and establishes a three-dimensional model through Solidworks software, and carries out a static analysis of the limit position in the operation process by using the finite element analysis software ANSYS Workbench, and obtains the maximum deformation under the limiting conditions of 0.1129mm, and the maximum stress value of 24.236MPa. The maximum deformation is 0.1129mm and the maximum stress is 24.236MPa. Subsequently, modal analysis is carried out to extract the first 6 orders of intrinsic frequency and corresponding vibration pattern, and harmonic response analysis is carried out on the weak components on the basis of modal analysis, which verifies that the structure complies with the design requirements in terms of strength, stiffness and resonance of the whole machine, and it provides the theoretical basis for the subsequent structural improvement and optimization design.

Keywords:Automatic loading and unloading; Vertical broaching machine; Manipulator; Structural design; Finite element analysis

1. INTRODUCTION

With the continuous development of China's modern manufacturing technology, the degree of automation in manufacturing industry is also increasing, as a direct actuator of automated production, the manipulator can complete the handling of objects, gripping and other functions. In recent years, domestic experts and scholars have done a lot of analysis and research on the design of material grasping automation manipulator, Sun Rui et al [1] in order to realize the automated handling operations on the packaging production line, established a three-dimensional structure model of the loading and unloading of man-made boards, and through the system dynamics simulation to derive the dynamic characteristics of the manipulator and verified the accuracy of the model. Zhou Yifeng [2] designed a press loading and unloading device in order to meet the modernization and development needs of the stamping method, and optimized the design of the core components. Xu Aiqun et al. [3] designed a truss manipulator for loading and unloading of tooth blanks on two CNC machines with different machining requirements in order to meet the actual working needs of a tooth blank CNC machining production line, and optimized the design of the truss manipulator with ANSYS Workbench software, which is the most suitable for the automated handling operation on the packaging production line. Workbench software to carry out static analysis of the limit position of the truss manipulator during the working process, to verify the reliability of the structure, and modal analysis of the truss manipulator.Bian R P [4] et al. used component counting method and fuzzy expert scoring method to analyze the reliability prediction and fault parts of the underfeeding truss robot on CNC punching machine, and verified the validity of the reliability prediction method and the accuracy of the prediction results. Tian Yuan et al. [5] for a hard tobacco filter rods of the composite additive demand, the design of a tobacco filter rods loading and unloading mechanism, the loading and unloading mechanism to carry out transient dynamics simulation, and from the point of view of structural dynamics to carry out dynamic characterization of the mechanism, to verify that the filter rods of the composite addition demand. The transient dynamics simulation of the loading and unloading mechanism is carried out, and the dynamic characteristics of the mechanism are investigated from the perspective of structural dynamics to verify the rationality and stability of the filter rod loading and unloading mechanism. Qi L [6] et al. designed a milling machine and loading device suitable for the forming of ring-shaped rotary parts, which can realize the automated roll forming of ring parts.

Based on the above research, for the actual situation of broaching machine processing, an automatic loading and unloading gripping robot for broaching machine processing is designed, which is simple in structure, reliable in performance, and can effectively improve the processing efficiency by combining with broaching machine, and use the finite element analysis software ANSYS Workbench to carry out static analysis and verify the reliability of the structure, and carry out modal analysis and harmonic response analysis of the structure under the limiting working condition, and come up with the band where the resonance is likely to occur, so as to provide theoretical references for the subsequent structural improvement and optimization.

2. LOADING AND UNLOADING ROBOT OVERALL STRUCTURE DESIGN
2.1 Robotic Work Layout Program

Through the analysis of the characteristics of workpiece processing and process flow, combined with the production needs, the entire production process of feeding, loading, processing, discharging, conveying to the storage of materials is determined. In order to achieve accurate and rapid linkage between the robot and the machine tool, to realize the automatic clamping, moving, placing of the workpiece, and to quickly complete the workpiece conveying and storing, the relative position of each executive part is arranged. Automatic loading and unloading layout plan is shown in Fig.2-1.

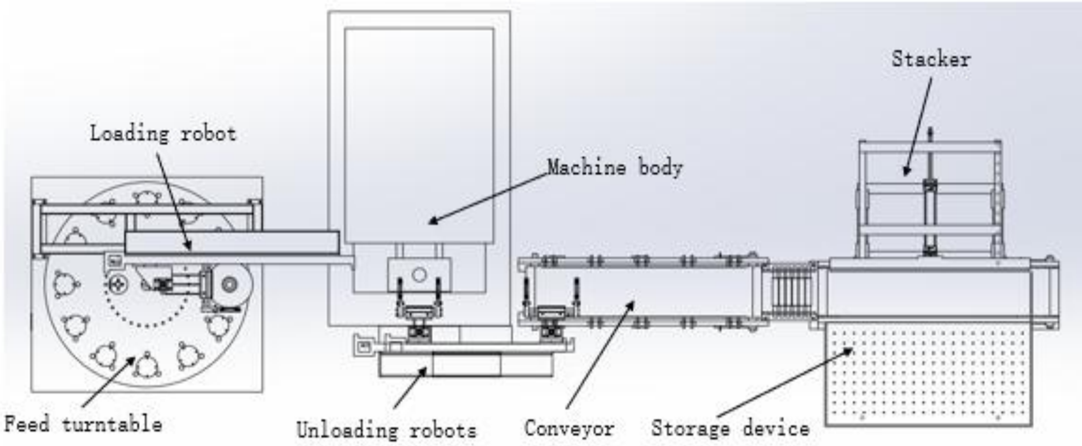


Fig.2-1.Automatic loading and unloading layout scheme plan diagram

2.2 Robotic Workflow

By comparing and analyzing the advantages and disadvantages of various industrial manipulator coordinate forms as well as considering the space utilization rate, the convenience of installation and the simplicity of action [7], the coordinate form of the manipulator designed this time is the right-angle coordinate type, and the mode of action is selected as linear movement. The overall structure of the manipulator includes the base bearing module, horizontal traverse module, vertical lifting module, end-effector module, and the modules cooperate with each other to complete the workpiece loading and unloading work. The three-dimensional model of the robot is shown in Fig.2-2.

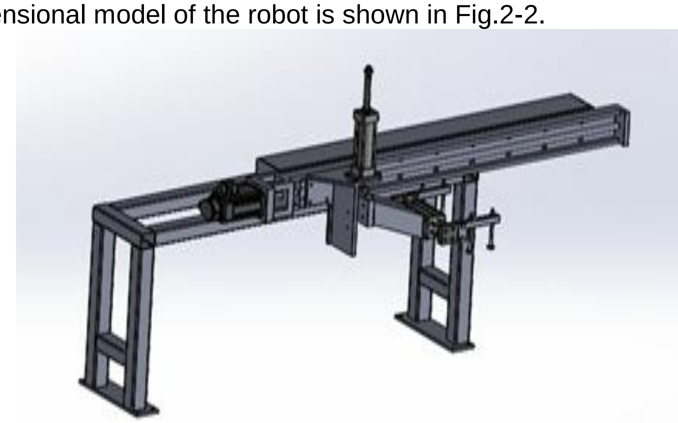


Fig.2-2. 3D model of the manipulator

The robot designed in this paper is mainly used for processing the loading and unloading process of the parts shown in Fig.2-3, and the state of the grasping blank is shown in Fig.2-4.

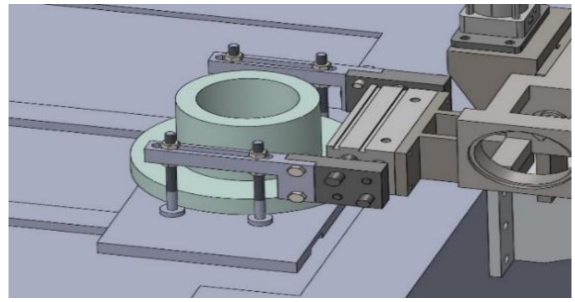


Fig.2-3. Parts to be machined Fig.2-4. Schematic diagram of gripping blank

3.SIMULATION ANALYSIS OF THE MANIPULATOR

As the manipulator in the operation process of the horizontal and vertical modules need to constantly rise - horizontal movement - descending and other actions, and need to withstand the workpiece, module and its own gravity, the force is constantly changing, different working positions will have a certain impact on the stiffness and strength of the manipulator, if the amount of deformation exceeds the allowable range (allowable error of ± 1mm), will affect the movement of the manipulator positioning accuracy. Therefore, it is necessary to focus on considering the rigidity and strength of the body at the limit position, as well as whether the whole machine will resonate during operation, to verify whether it meets the requirements of the machining task, so as to judge whether the structure meets the design requirements, and to avoid the phenomena of low precision, insufficient output and even structural damage during actual machining [8]. In the following, ANSYS Workbench is used to carry out finite element analysis of the manipulator. Because the loading and unloading manipulator structure and function are similar, only the action process is opposite, the following only take the loading manipulator to do the analysis.

3.1Finite Element Modeling

As a complex assembly, the manipulator should be simplified appropriately before conducting finite element analysis [9], removing unimportant features such as chamfers, threads, etc., and converting the complex model of the drive motor, drag chain and pallet in the module into a load to be loaded into the corresponding position. The simplified model is shown in Fig.3-1.

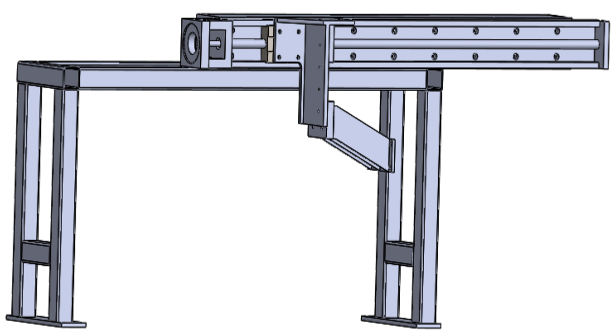


Fig.3-1. Simplified model diagram

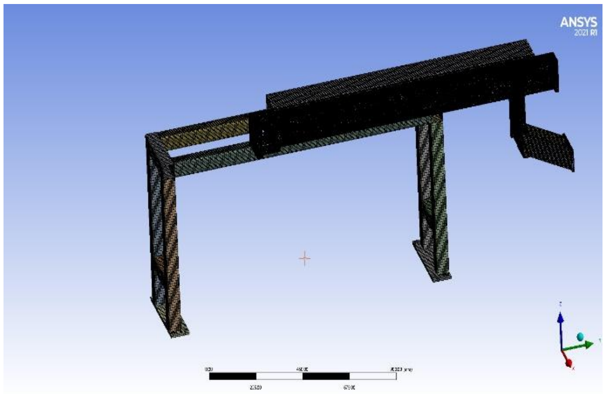
3.2Static Finite Element Analysis

Before finite element analysis of the manipulator, the corresponding material type and related properties need to be set. The main material of the manipulator is Q235 steel, including the support frame, cylinder bracket and actuator bracket, and the material of the guideway module is set to 45 steel. The material properties are shown in Table 1.

Table 1. Material properties

Materials	Density/(g.cm^3)	Elastic Modulus/GPa	Poisson's ratio	yield Strength/Mpa
Q235 steel	7.85	210	0.270	235
45 steel	7.85	209	0.269	355

After setting up the material properties, the model is meshed, and the relationship between the mesh quality and the simulation results is very close, in order to effectively avoid the stress concentration phenomenon in the simulation process and improve the calculation speed, the free mesh delineation method is easy to get a better mesh quality [10], and the X-axis translation module is connected with the support frame and the reclaiming bracket part of local encryption, and the cell size is set as 3mm to improve the mesh quality. The final meshing statistics have 4766631 nodes and 2832124 cells, and the average value of its skewness is 0.204, which indicates that the meshing quality is very good. The analytical model after meshing is shown in Fig.3-2.



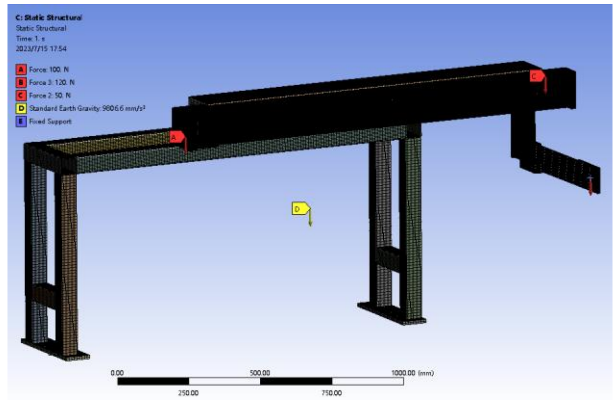
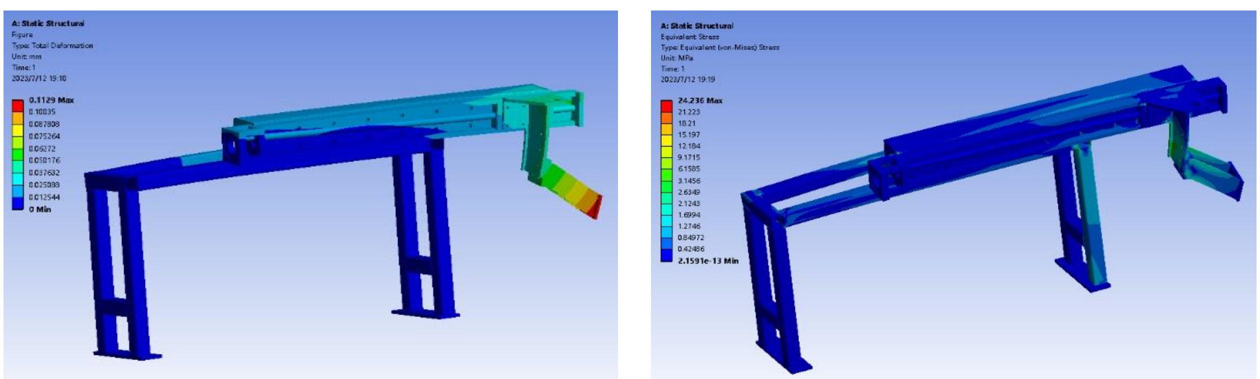


Fig.3-2. Meshing of the model   Fig.3-3. Manipulator load force diagram

Fixed constraints and loads are added to the manipulator after meshing. The manipulator should be fixedly mounted on the base of the feeding turntable, Fixed constraints are added at the base position of the two brackets of the manipulator, due to the simplification of the motor and other components when processing the model, a vertically downward load of 100N is added at the end of the transverse module, and a vertically downward load of 50N is added at the cylinder bracket, the mass of the pickup gripper module and the workpiece are 2kg and 10kg respectively, so the maximum load condition of the manipulator is 12kg, and 120N is applied at the front of the actuator end bracket. The maximum load condition of the manipulator is 12kg, and a force of 120N is applied to the front end of the actuator end bracket, and finally a simulated gravity field in the negative direction of the Z-axis of the space is added. After loading each part of the load and force is shown in Fig.3-3.

Subsequently, the hydrostatic solution analysis was carried out to obtain the deformation cloud and equivalent stress cloud as shown in Fig.3-4.



（a）clouds of deformation（b）stress map

Fig.3-4. Cloud diagram of deformation and stress distribution of the manipulator

Viewing the analysis results in Fig.3-4(a), it can be seen that the maximum deformation occurs at the end-effector bracket, and the deformation is 0.1129mm, which meets the engineering requirements of controlling the deformation within ±1mm, and has little impact on the overall structure. From Fig.3-4(b), it can be seen that the maximum stress occurs in the support assembly of the manipulator, and the maximum stress value is 24.236MPa, which is much lower than the yield strength of the material used for the support frame. The yield strength of the material used for the support frame is much lower than the yield strength of the material used for the support frame. The analysis shows that the structure of the manipulator meets the performance requirements, and there is a large redundancy in the deformation, strength and stiffness of the structure, which has a lot of space for optimization.

3.3 Dynamic Finite Element Analysis

3.3.1 Modal analysis

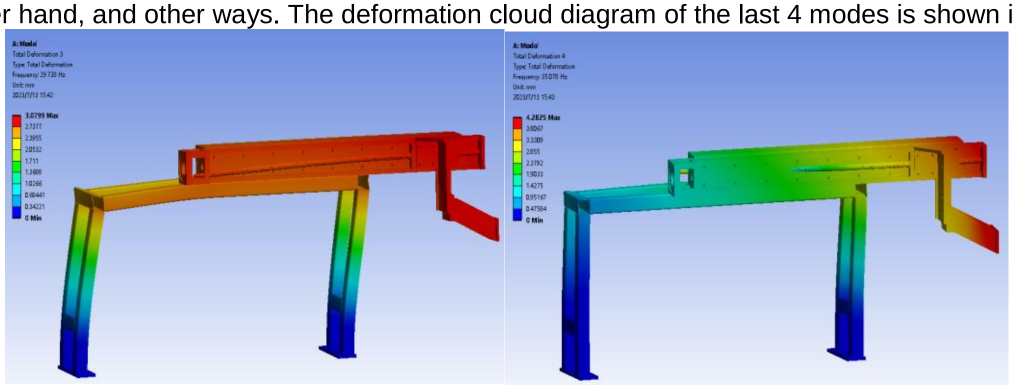
Modal analysis is to solve the structural vibration equations of the research object, as an effective means to study the dynamic characteristics of the structure, can obtain the mechanical structure in a certain frequency range of each order of

the intrinsic frequency, the corresponding vibration pattern and other parameters, so as to reflect the weak parts of the structure [11], for the structural optimization of the manipulator's design to provide important parameters.
The modal analysis module is added on the basis of static structural analysis, and the Subspace method is used to calculate the modal analysis. From the mechanical vibration theory, it is known that in the process of structural vibration, low-order intrinsic frequency corresponds to the main role of the vibration mode, so the first 6 orders of intrinsic frequency and vibration mode are extracted. The first 6 orders of modal intrinsic frequencies and shapes are shown in Table 2.

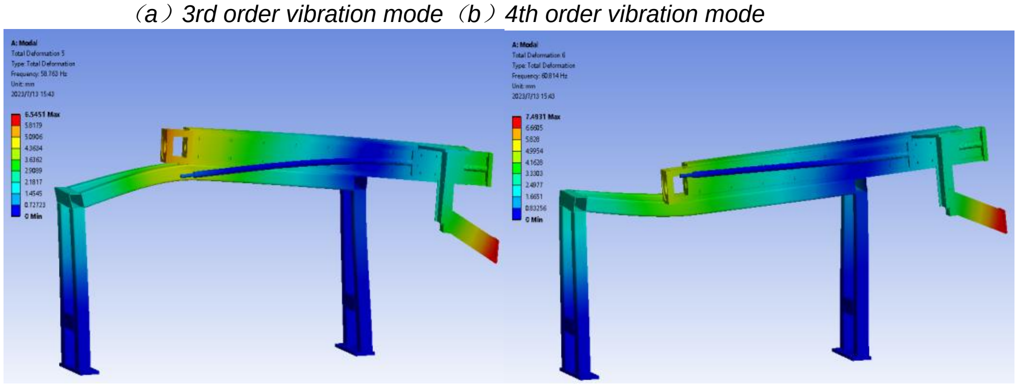
Table 2 First 6 orders of intrinsic frequency and vibration pattern

Modal Order	Solid Frequency /Hz	Shape Description
1	15.19	Left and right oscillations along the X-axis
2	15.401	Oscillation back and forth along the Y-axis
3	29.728	Twisting in the YOZ plane
4	35.078	Twisting in the XOZ plane
5	58.763	Oscillation up and down along the Z-axis
6	60.814	Twisting in the XOY plane

The 1st order mode reflects the horizontal vibration of the manipulator along the X-axis, the 2nd order mode reflects the vibration in the horizontal direction of the Y-axis, the 3rd, 4th and 6th order modes reflect the torsional vibration of the manipulator in the space, and the 5th order mode reflects the longitudinal vibration of the manipulator in the Z-axis, which is mainly due to the vibration generated by the Z-axis lifting cylinder starting, braking, and lifting of the workpiece by the mechanical gripper hand, and other ways. The deformation cloud diagram of the last 4 modes is shown in Fig.3-5.



（a）3rd order vibration mode （b）4th order vibration mode



（c）5th order vibration mode （d）6th order vibration mode

Fig.3-5. Modal vibration pattern cloud

From Table 2 and Fig.3-5, it can be seen that the first 6 orders of the intrinsic frequency between 15~61Hz, the overall structure of the low-order vibration pattern is mainly the vibration of the support frame, the translation module and the end-effector bracket, especially in the 6th order vibration pattern, the end-effector bracket deformation of the largest amount of 7.49mm, in order to avoid the structural deformation of the manipulator and the machine tool and the worktable and so on to interfere with the material loading and unloading failures, the later operation control should be strongly avoided in this band. In order to avoid the failure of loading and unloading due to the interference between the robot and the machine body and table caused by the structural deformation, this band should be avoided in the later operation control.

3.3.2 Harmonic response analysis

Harmonic response analysis mainly reflects the dynamic response of the structure when subjected to different sinusoidal law changes in load, so as to verify whether the structural design can overcome the system resonance and fatigue and other problems[12], the manipulator structure for harmonic response analysis.
Add the harmonic response module on the basis of modal solving, use the Mode Superposition method to analyze the harmonic response of the manipulator[13], according to the results of modal analysis and the actual working conditions to set the frequency range of 0~60Hz, the frequency interval is set to 1Hz, and carry out the calculation of the solution.

The overall deformation of the manipulator after the harmonic response analysis solution is shown in Fig.3-6, and the deformation of the structure is mainly concentrated in the end-effector bracket, and the deformation results of the bracket in each direction are derived to obtain the displacement response curves in each direction shown in Fig.3-7.

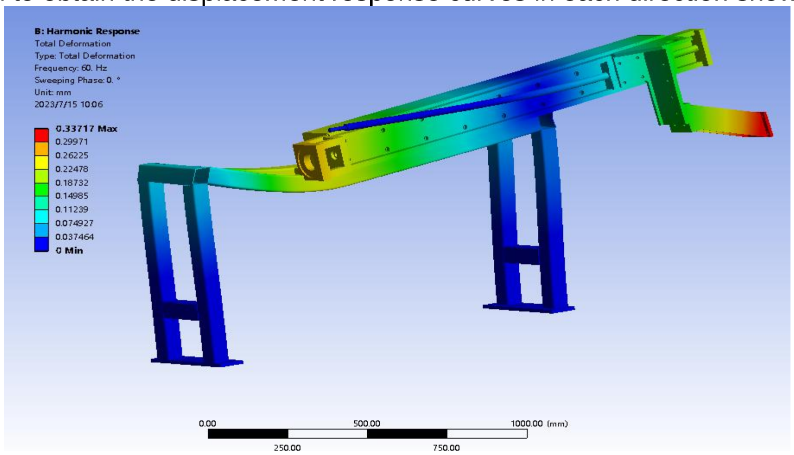
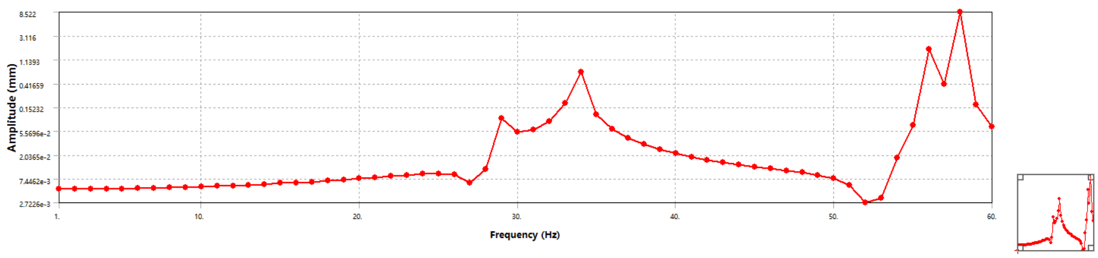
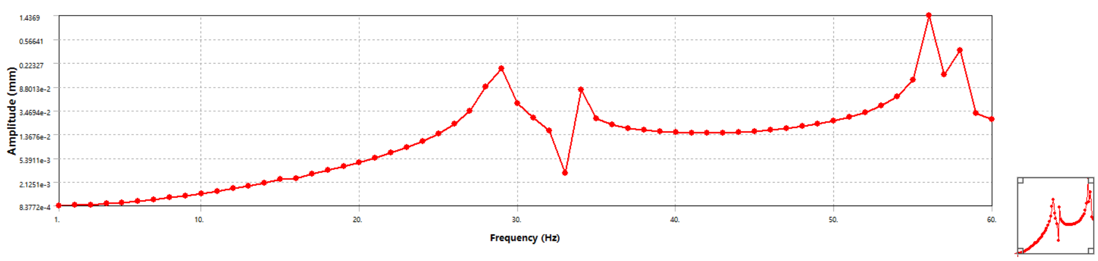


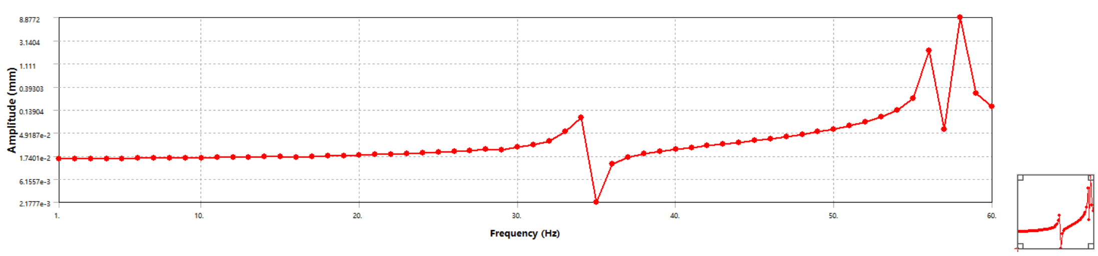
Fig.3-6. Cloud diagram of the overall deformation of the manipulator



(a) X-direction



(b) Y-direction



(c) Z-direction

Fig.3-7. Displacement response curve of the support frame in each direction

Comparing the displacement response values of each direction under different frequencies, it can be seen that the deformation of the support frame in each direction of X, Y and Z is mainly concentrated in the frequency bands of 26~35Hz and 53~60Hz, and the deformation of the X direction in this frequency band floats more, with a maximum deformation value of 0.52mm, and the deformation of the Y direction mainly occurs at 30Hz, with the maximum deformation in the frequency band range of 0.145mm, which is a smaller deformation, and the Z direction has the largest deformation float in the band range. The Z-direction has the largest deformation float in the band range, and the maximum deformation of 0.55mm occurs at 57Hz. The above analysis shows that when the manipulator operates in the frequency band range, the end-effector bracket mounted on the X-axis traversing module will generate positioning accuracy errors due to resonance, and the frequency band in which the deformation occurs contains several orders of the intrinsic frequency obtained from the modal analysis. Although the amount of deformation generated is small, it has a large impact on the positional accuracy when working in the X and Z directions, so it should be avoided as much as possible when working.

4.CONCLUSION

In this paper, through the low production efficiency of a company's broaching machine processing, the labor intensity of workers and the need to improve the efficiency of the reality of the problem, designed to replace the manual loading and unloading of the manipulator, through the determination of the overall production layout, structural composition and analysis of the key execution components to verify. Based on ANSYS Workbench, the static analysis and modal analysis are carried out on the limit position of the structure during operation, and the harmonic response analysis is carried out on the bracket of the end-effector considering the impact of motor vibration and other loads on the bracket. The analysis results show that the stiffness and strength meet the design requirements, fully meet the workpiece loading and unloading operations, and the analysis concluded that the actuator bracket in the 26~35Hz and 53~60Hz operation, the deformation in all directions is the largest, easy due to resonance causes errors in precision or even interference with the machine tool, is not conducive to the stable operation of the system, the actual operation of the frequency band should be avoided. The research content of this paper can provide theoretical reference for the subsequent structure optimization and improvement of design.

COMPETING INTERESTS

Authors have declared that no competing interests exist.

AUTHORS' CONTRIBUTIONS

This work was carried out in collaboration among all authors. All authors read and approved the final manuscript.

REFERENCES

1. Sun R, Xue B, Wang X, et al. Dynamic analysis and simulation of loading and unloading manipulator for artificial board[J]. Packaging Engineering,2022,43(17):130-139.
2. Zhou Yifeng. Structural design of unloading robot on press[J]. Forging Equipment and Manufacturing Technology,2021,56(06):31-34.
3. XU Ai-Qun, RUI Hong-Ye. Structural design and analysis of truss manipulator for loading and unloading of tooth blank[J]. Journal of Zhejiang Institute of Science and Technology,2023,35(03):234-242+258.
4. BIAN Renpeng,JIAZhixin,HEKotai. Reliability prediction and validation of a truss robot for unloading on CNC punching machine[J/OL]. Journal of Harbin Engineering University:1-10.
5. Tian Yuan, Zhong Fei, Yang Xiaojun et al. Design and finite element analysis of loading and unloading mechanism of cigarette filter rod[J]. Packaging and Food Machinery,2023,41(01):80-85.
6. QI Lei, GUAN Yabin, LIU Jun et al. CNC horizontal ring rolling machine and automatic loading and unloading device[J]. Mechanical Design,2023,40(S1):143-146.
7. Song Bei, Luo Jing, Xiao Hang et al. Automated production line for internal spline broaching process of cam parts[J]. Manufacturing Technology and Machine Tools,2019,No.686(08):164-168.
8. XuWei,ZhuHuaizhi,ZhongYuchao et al. Structural optimization design of loading and unloading manipulator for hot pressing system based on NSGA-Ⅱ[J]. Modern Manufacturing Engineering,2023,No.512(05):65-73.
9. LiYanan,ZhangWei,CaoDongxing,et al. Modal experiment and finite element simulation analysis of ring truss structure[J]. Journal of Applied Mechanics,2017,34(01):119-124+200.
10. ZHU He,HUANGChenlei,YANG Liming et al. Optimization of quadcopter UAV frame structure based on response surface method and topology optimization[J]. Mechanical Design,2023,40(S2):130-135.
11. Xie Jialiang,ZhangChunliang,Zhu Jianguo et al. Dynamic and static characteristics and structural optimization of gantry manipulator[J]. Machine Tools and Hydraulics,2018,46(09):65-70.
12. HU Yibo, GAO Zicheng, LI Lijun et al. Research on vibration characteristics of insertion machine based on linear motor drive[J]. Manufacturing Technology and Machine Tools,2023(12):32-39.
13. LIU Yang,XINGJianguo,WANGLongjie. Design and optimization of handling manipulator based on Pro/E and Ansys Workbench[J]. Journal of Qingdao University (Engineering and Technology Edition),2020,35(01):45-48+64.