Finite Element Analysis for Cartesian Robot with Triple-speed Truss

Junling Zhang, Jirong Wang, and Mingshun Su
Mechanical and Electrical Engineering Institute, Qingdao University, qingdao 266071, China
Email: 15764219179@163.com, wangjirong43@163.com

Jun Li
School of Data Science and Software Engineering, Qingdao University, qingdao 266071, China

Guangtao Liu
SIASUN Robot & Automation CO., LTD, qingdao, China

***Abstract*—Cartesian robot with triple-speed truss can achieve the function that was grabbed tire from an assembly line and completed tire vertical stacking of storage. This paper was mainly focused on the design and parameter optimization of triple-speed truss robot-arm. The virtual prototype of Cartesian robot was built by Solidworks software. Using of finite element analysis software ANSYS Workbench, key parts and robot in the system were static analyzed and modal analyzed so that product was designed to meet the strength and stiffness requirements.**

***Index Terms*—Cartesian robot with triple-speed truss, finite element static analysis, modal analysis**

I. Introduction

It is understood that industrial robot technology has been widely used in foreign tire business logistics and intelligent production. Michelin, Bridgestone, Pirelli and other tire companies have a unique intelligent production technology so that the global tire manufacturing industry has the trend of intelligent development. More and more industrial robots are also being put into use in the Chinese tire industry [1]. China's industrial robots have made great progress; the market has gradually matured [2]. China has a large number of enterprises and scientific research workers in the innovative research and development of industrial robot technology for the international forefront of high-tech. Xiaoqing Li of [3] Guangxi University designed a multi-degree-of-freedom controllable palletizing robot based on modern organization's innovative design theory and method. And the motion analysis and simulation of the mechanism were carried out by using Pro / E and ADAMS / View. Improve the movement characteristics and dynamic characteristics of the organization, can better achieve high speed, accurate, reliable and continuous palletizing operations. Chengwei Li [4] etc. designed a four-degree-of-freedom palletizing robot for the logistics automation industry based on the performance requirements of handling robots. The use of distributed secondary control structure to achieve system monitoring and job management, accurate tracking trajectory planning and independent development of the palletizing robot control software. Jinfang Xia [5] of Qingdao University of Science and Technology designed automatic conveying system for all-steel radial tire. A set of mechanical equipment for automatic conveying of all-steel radial tire including flip manipulator and molding gantry robot is designed. Robot of main task is to transport all-steel radial tire to the tooling plate speed chain. It is a great significance to improve tire production efficiency, reduce labor intensity and realize high speed, high efficiency and high quality tire transportation. The use of Cartesian Robot with Triple-speed Truss can improve automated production, reduce the labor intensity of workers, improve labor productivity, and significantly improve economic efficiency.

The disadvantage of the existing technology is that the y-axis stretching length requires longer time-doubled speed chain which can not be realized. The grasping manipulator uses two servo systems and pneumatic systems, which have high cost and low efficiency. The aim of the present invention is to design a triple-speed truss manipulator system to overcome the shortcoming of short Y-axis extension length. The grasping manipulator uses two servo systems and grasping mechanism with low cost and high efficiency.

In this paper, the static analysis and modal analysis of cartesian robot are performed based on finite element analysis. The distributing nephogram of stress and strain and the first six modes shape are obtained to ensure product can meet the strength requirements.

Manuscript received December 1, 2018; revised December 14, 2018.

doi: 10.18178/joace.6.2.57-60

II. Statics Analysis of Horizontal Drive Motor Mount

Under actual operating conditions, the level-driven motor mount is required to withstand half the total load in the vertical direction. The installation of the motor requires a threaded hole and the size of the center hole fitted to the motor. So the need for static analysis of the mounting bracket to ensure that it will not be destroyed in the actual work process. The dongle model obtained by Solidworks modeling is shown in "Fig. 1."

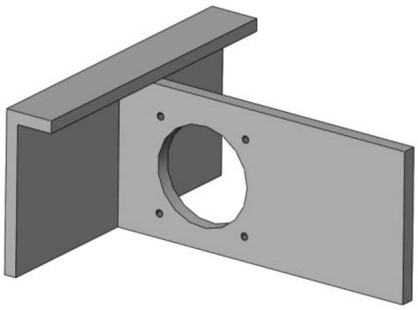

Figure 1. Work platform diagram

Import the model into the Ansys Workbench for static analysis. Specific steps are as follows. Establish a static analysis project

Drag or double-click Static Structural in the toolbar to create a static analysis project on the right side of the project icon area [6] [7].

A. Set the Material and Its Properties

Enter the Engineering Data to set up the mounting material for Structural Steel, density $\rho=7.85\times10^{-3}$ $Kg\cdot mm^3$, elastic modulus $E=2\times10^{11}$ Pa, Poisson's ratio $\mu=0.3$.

B. Model Import and Meshing

Import the 3D model shown in Fig. 1 into the Workbench, add the Method in the Mesh option, set the meshing method as Multizone, set Relevance to the maximum of 100 in the meshing list and set the Relevance Center to Fine. Finally, the grid is divided, resulting in 229841 Nodes, 48049 Elements, as shown in "Fig. 2."

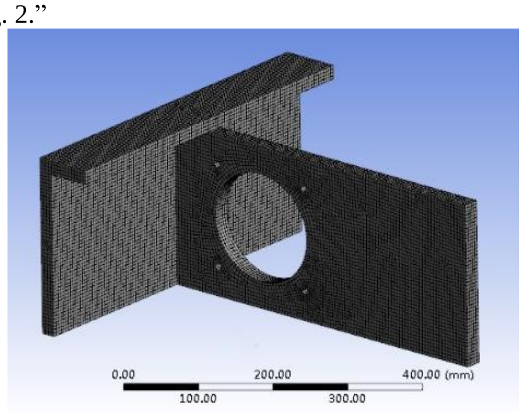

Figure 2. The grid of horizontal drive motor mount

C. Add Constraints and Force

The left end face of the mounting bracket is welded with the moving body bracket as a whole in the actual working condition. The motor is connected to the mounting bracket via the right four screw holes. Therefore, here apply a constraint to the left end of the face of the application of Fixed Support, in the four screw holes applied to the size of 7155.162N and along the-z direction of the center circle center 43.5mm Remote Force, the direction of Y positive. Finally, the solution is calculated.

D. Post-processing

Add Total Deformation to Solution to get the deformation of the mounting bracket. At the same time, add the user to customize the results of the project, due to the material used Q235A plastic material, according to the theory of distortion energy density (Fourth strength theory) [8], enter in Expression: $S1\text{-}b\times S3$, $b=\sigma_\tau/\sigma_c$, σ_τ the allowable tension and σ_c the allowable compressive stress for two stresses of carbon steel are equal. Finally get the deformation cloud and stress cloud as shown in "Fig. 3" and "Fig. 4."

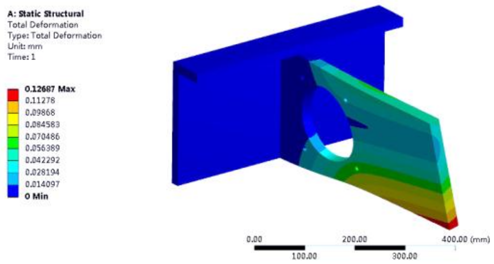

Figure 3. The deformation cloud of horizontal drive motor mount

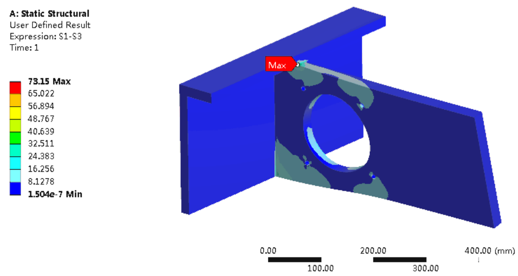

Figure 4. The stress cloud of horizontal drive motor mount

As can be seen from the above data, the maximum displacement of the horizontal drive motor mounting bracket is 0.12687mm, which appears in the red position at the right end of the mounting bracket. According to the theory of distortion energy density, the maximum stress is 73.15MPa, which occurs at the position indicated by the red icon Max in the figure.

III. Modal Analysis of Machine Handle

There are some vibrations in the movement of the robot assembly, and the natural frequencies of the components are different. Modal analysis of the robot assembly is carried out in order to allow the overall system to operate stably without resonance and other vibrations.

As the machine hand-mounted components contain more parts, if the actual situation in accordance with the analysis needs to spend a lot of time and resources. The model will be simplified in order to use the least resources and the highest efficiency to solve the practical problems in the actual engineering analysis. Before modal analysis of the robot assembly, the model is simplified in this paper, including:

A. Simplified Treatment of Aluminum

Due to the fact that the aluminum profile is more complex, the actual model can be used to achieve the best results in the analysis of individual part. However, in the assembly, due to the complex cross-section will be derived from other unnecessary question, this paper will be simplified aluminum section, with a similar structure of the rectangular tube instead of the analysis of the assembly.

B. Process the Solidworks Model

Because Solidworks software has strong flexibility in the modeling process, the same part can be modeled with different processing methods. But in the ANSYS Workbench for the import of Solidworks model identification and Solidworks is not the same processing logic. The model is boolean to merge unwanted entities before the analysis.

C. Compression Processing

Modal analysis of the body of robot assembly is in the state where the robot assembly does not undergo lateral displacement. The robot assembly is stationary in the horizontal direction, only the speed chain and the robot move up and down. Also, in this state, compression is performed for each non-stressed component to simplify the operation.

In the process of mesh partitioning, contact relationship remains default, but make sure that no Bodies or Parts will appear in options "Bodies Without Contact in Tree" and "Parts Without Contact in Tree", otherwise the Body and Parts listed in the design tree should be adjusted. In the grid setting option, set Relevance to 60, Relevance Center to Medium, and Smoothing to Medium. Finally, 474481 Nodes and 113564 Elements are obtained by meshing, as shown in “Fig. 5.”.

The manipulator is a prestressed modal analysis for modal analysis. At first the static analysis on the model should be done [9]. The specific steps of the static analysis are the same as those described in the previous section. Perform modal analysis. Set the order of the modal analysis to 6 order. Finally, the results of the first six orders of modal shape are shown in “Fig. 6.1.-6.6.”.

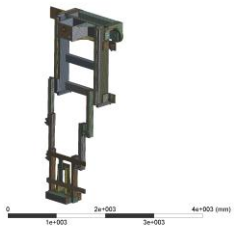

Figure 5. Meshing of Robot Assembly

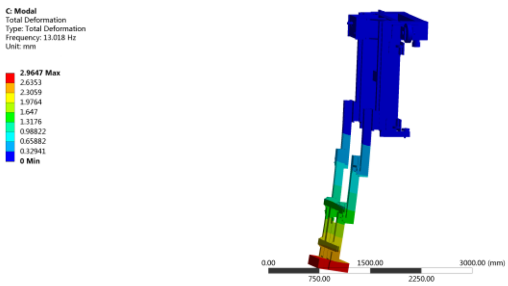

Figure 6.1. First-order mode shapes of robot assembly

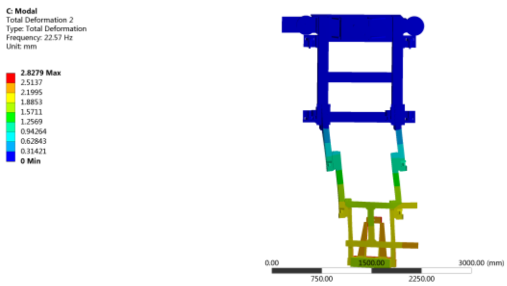

Figure 6.2. Second-order mode shapes of robot assembly

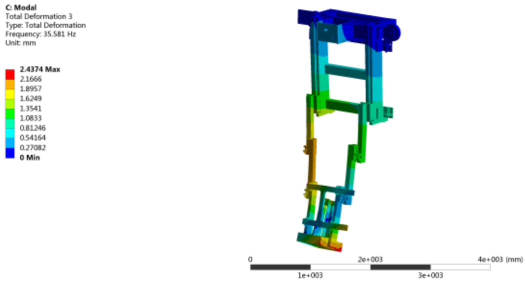

Figure 6.3. Third-order mode shapes of robot assembly

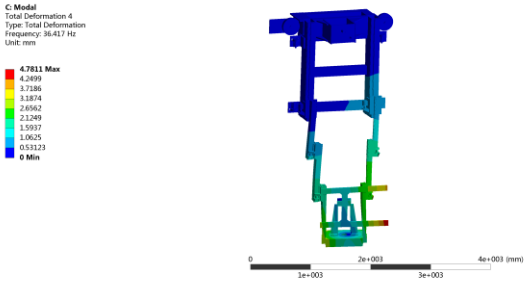

Figure 6.4. Fourth-order mode shapes of robot assembly

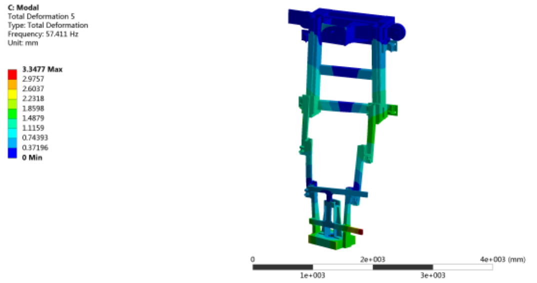

Figure 6.5. Fifth-order mode shapes of robot assembly

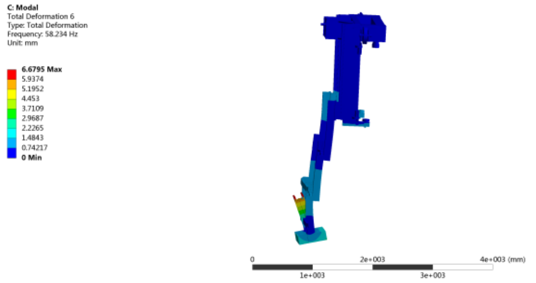

Figure 6.6. Sixth-order mode shapes of robot assembly

By looking at the modal vibration frequency data sheet, the natural frequencies of the robot assembly in 1~6 order are: 13.018Hz, 22.57Hz, 35.581Hz, 36.417Hz, 57.411Hz,58.234Hz. The vibration source in this robot assembly comes from the servo motor. The Mitsubishi servo motor to provide the mechanical resonance filter frequency range of 10HZ ~ 4500HZ, can use the maximum number of filters for the five. In this case, the natural frequency of the first six orders of the robot assembly is about 60 Hz, and it is within the operating frequency range of the mechanical resonant filter. Therefore, it is possible to avoid possible vibration in actual use.

IV. Conclusion

In this paper, the key components were analyzed by the finite element analysis software ANSYS Workbench. Maximum deformation cloud image and maximum stress cloud diagram of the designed part is obtained. Comparing the allowable stress of the material with the use of the material, the strength and stiffness requirements of the designed parts are verified. At the same time, modal analysis of the robot assembly was carried out. According to obtain by the natural frequency of comparison in the normal operation period does not occur resonance phenomenon to ensure the reliable operation of the equipment.

References

[1] "China will become the world's largest industrial robot market in 2015." [Online]. Available: *http://www.chinairn.com/news/20140909/094828752.shtml.*

[2] P. Wang and L. Y. Zhu, "Motion simulation of grab manipulator and finite element analysis of its waist," *Journal of Mechanical Transmission*, vol. 35, no. 5, pp. 43-45, May 2011.

[3] X. Q. Li. "Kinematic analysis and simulation of multi-freedom palletizing robot with controllable mechanism," Guangxi University, 2013.

[4] C. W. Li, X. L. Zhu, and C. Yun. "Design and research of stacking robot kinematics," *Mechanical & Electrical Engineering Magazine*. vol. 25, no.12, pp. 81-84, Dec.2008.

[5] J. F. Xia. "Design and research of all steel radial tire automatic conveyor system," Qingdao University, 2013.

[6] B. Li and H. P. Gong, "ANSYS14 finite element analysis self-study manual," Beijing: People's Posts and Telecommunications Press. 2013, 4.

[7] J. F. Chen. "Finite element analysis of pneumatic dual-robot gantry device," Nanjing University of Science and Technology, 2002.

[8] H. W. Liu. "Material mechanics," Beijing: Higher Education Press. 2011, 1.

[9] M. Cong and H. L Shi, "Finite element analysis on dynamic characteristics of gantry robot," *Modular Machine Tool & Automatic Manufacturing Technique*, no. 6, pp. 19-22, June 2008.

Junling Zhang was born in 1993. She is studying for a master's degree in Qingdao University. Her research interests include the structure design of robot and finite element analysis.

Jirong Wang was born in 1967. She graduated from the Chongqing University and acquired the Bachelor of Engineering in 1987 and She graduated from the Ocean University of China and acquired the Ph.D.degree in 2009. Now She is an Professor in Qingdao University. Her research interests include teaching and research in mechanical design and robotics.

Mingshun Su was born in 1994. He graduated from the Qingdao University and acquired Master's Degree in Engineering. His research interests include the design of Automatic Production Line.

Jun Li was born in 1964. He is a doctor of engineering, a researcher, and a supervisor of master's degree students. His research interests include System Modeling and Simulation, System Dynamics and Computer Engineering Simulation.

Guangtao Liu was born in 1989. He graduated from the Qingdao University and acquired Master's Degree in Engineering. His research interests include the design and research of robot.