Finite Element Analysis of Pick and Place Robotic Structure

Vharkare C. B[1], Jagtap P. V[2], Gadade R. A.[3], Mate A. K.[4]

[1] *Lecturer, Mechanical Department, M. S. Polytechnic, Beed, Maharashtra, India*
[2] *HOD, Mechanical Department, Aditya Engineering college, Beed, Maharashtra, India*
[3] *HOD, Mechanical Department, M. S. Polytechnic, Beed, Maharashtra, India*
[4] *Lecturer, Mechanical Department, M. S. Polytechnic, Beed, Maharashtra, India*

ABSTRACT

The robot supporting structures play a very vital role in the motion of robots. The robot on a confined location such as ceiling or foundation can move in any direction. Due to this forces and moment act on the structure at the region where the mounting for robot is provided. The maximum deflection in the structure is at the support location. Also, the structure vibrates due to the natural frequency. If this frequency becomes equal to the frequency of other moving elements, the possibility of resonance increases. Hence to keep the structure from oscillating with maximum frequency the need of frequency optimization is necessary. The complete structure is kept in the limits of minimum deflection due to vibration by considering the harmonic response.

Keyword: - *Resonance, current, vibration, ANSYS*

---

1. Introduction

Robotic structures are those which are used to support the robot assembly. The structure provides complete support for the motion for robot. These structures are built according to the necessity of the robot. Here, the pick and place robot is supported with a pedestal which is ultimately attached with the overhead structure.

Optimized structural design for the structures of the industrial robots have to meet certain criteria regarding dimensional design and shape, material consumption and adapt this to the functional requirements. For an optimized design of the robot structure the engineers normally consider all the aspects of industrial applications where the structure will be integrated. Specific requirements are related to the resistance of the elements, not to oversize the structure but also to guarantee minimum criteria of stability and security in operation and to fit the material and its shape with the above mentioned criteria. It is required to correlate all these with the kinematic model of the joints and from this basis to establish the loads and to build a dynamic model to determine the behavior from this point of view. [1]

1.1 SCOPE

The availability of data required for complete design of the robotic structure is not properly categorized. There is a vast need of experimental data. This data is not available easily as many researchers are working on it. Assumptions of many parameters are to be considered while designing as the forces and motions are not particularly specified. There is always a confined scope in consideration of data.

1.2 METHODOLOGY

The brief idea about the steps carried out for the analysis and validation of the structure. The methodology is initiated with the design of the hollow section by using the data provided by the company. Similar structure was considered for trial models. The meshing and basic structural analysis was done on the trial model by applying boundary conditions to restrict the deformation within the given limits. Then the model with required deformation and stress value was considered for basic analysis. As the natural frequency of the structure is to be increased to a certain value to avoid resonance, the structure was optimized according to the provided conditions until the required frequency is obtained. After obtaining the required frequency, further harmonic response was calculated on the same structure. This result was validated with the experimental result.

2. HARMONIC ANALYSIS

There are some phenomena which exhibit a repetitive pattern, e.g., a sine wave, motion of a pendulum, etc. To solve such kind of problems so as to get answers to questions like what is the magnitude of maximum and minimum point, frequency of repetition, etc., a typical type of solution method has to be implemented. Harmonic analysis is one such mathematical procedure for describing and analyzing phenomena of a periodically recurrent nature. Many complex problems having a complicated mathematical equations and in turn complex curves have been reduced to manageable terms by the technique of breaking them into sums of comparatively simple components. This can be approximated to a simple mathematical formulation which states that a circle with infinitely large diameter can be assumed to be a straight line. On similar lines, a complex curve can be broken into small and simple curves with some approximation and then this can be solved.

As mentioned above, many physical phenomena, such as sound waves, alternating electric currents, tides, and machine motions and vibrations, may be periodic in character. The variables of measurement of such periodic motions are a dependent variable, measured with respect to an independent variable, usually, time. A

considerable amount of data is collected for a number of successive values for dependent and independent variables and a curve is plotted from them. This will represent a function in terms of the independent variable. Generally, it is observed that the mathematical expression for the function will be unknown. However, with the periodic functions found in nature, the function can be expressed in terms of trigonometric terms such as the sum of a number of sine and cosine terms. Such a sum is known as a Fourier series, after the French mathematician Joseph Fourier (1768–1830).The determination of the coefficients of these terms is called harmonic analysis. The term in this Fourier series that has a period equal to that of the function, f(x), is called the fundamental. Other terms are integral submultiples of the fundamental and have shortened periods are called harmonics. This terminology is derived from one of the earliest applications, the study of the sound waves created by a violin.

3. CONCLUSIONS

As the robotic structure was considered, the design was completely according to the company requirement. The trials were conducted to finalize model for deformation below the range of safety. The final trial model was considered for optimization on frequency. Optimization was further carried out in the geometry to increase the frequency. The frequency was increased up to required limit. The final model obtained after all the cases was initiated for harmonic response analysis for the required range of frequency in confined number of intervals.

The deformation of harmonic response for the range of frequency results to an approximate value. This value is then validated with the experimental test results obtained from company. The error for accuracy is calculated to check for the reliability of the structure.

4. ACKNOWLEDGEMENT

It gives me great pleasure to submit my Project report on **“Finite Element Analysis of Pick and Place Robot Structure”** to fulfillment of my ME (CAD CAM). I take this opportunity to thank my Project guide **Prof. P. V. Jagtap** for his valuable guidance and his deep interest throughout the study and completion of the Project.

It would be befitting to mention a word of thanks to **Dr. P. S. Kadu**, Principal Aditya Engineering College,Beed.

They have helped me through their expertise in their fields along with their invaluable advice helped me understand the basics of dynamics and its applications in the real life problems. I would also like to take this opportunity to express my gratitude towards the staff members of the department for their continuous support.

5. REFERENCES

[1].Grover G. K., Mechanical Vibrations, 8th ed., Nem Chand & Bros, Roorkee, U. K., India, pp. 1 – 2.

[2].Bhandari V. B., Design of Machine Elements, 3rd ed., McGraw Hill Education (India) Private Limited New Delhi, pp. 81 – 82.

[3].Bathe K. J., Finite Element Procedures. Prentice Hall, 2001, pp. 1-14.

[4]. Gokhale N. S., Deshpande S. S., Bedekar S. V., Thite A. N., Practical Finite Element Analysis, 1st ed., Finite To Infinite Pune, pp. 49-60.

[5]. S. Ramamrutham, R. Narayanan, Strength of material, 14th ed., Dhanpat Rai publication co., pp. 217-230.

[6]. Cook, R.D., Malkus, D.S., Plesha, M.E., Concepts and Applications of Finite Element Analysis, John Wiley and Sons, 1989. pp 1-15.
[7]. Zhijun Wu, Kailiang Lu, Huiqing Qiu, Finite Element Comparative Analysis of Two Doorframe Structures in Container Crane, National 863 plans projects. No.2009AA043000, IEEE Journal 2011, pp. 302 – 305.
[8]. Jaydeep Roy and Louis L. Whitcomb, Structural Design Optimization and Comparative Analysis of a New High-Performance Robot Arm via Finite Element Analysis, IEEE Journal 1997, pp. 2190 – 2197.
[9]. Masahide Matsumoto, Etsuo Iwaya, Interval Finite Element Analysis to Structural Systems, Proceedings of the 2000 IEEE Journal, pp. 132 – 136.
[10]. Gwang-Jo Chung, Doo-Hyung Kim, Structural Analysis of 600Kgf Heavy Duty Handling Robot, IEEE Journal 2010, pp. 40 – 45.
[11]. Xiaoping Liao, Changliang Gong, Yizhong Lin, Weidong Wang, The Finite Element Modal Analysis of the Base of Welding Robot, IEEE Journal 2010, pp. V2-123 – V2-126.