Finite element analysis of pick and place robotic structure

Shruti Udameeshi [1], Gayatri S Patil [2], Mr. Vinaay Patil [3]

[1] *Scholar, ME Design, Department of Mechanical Engineering, K.J College of Engineering & Management Research, Pune, Maharashtra, India*

[2] *Assistant Professor, Department of Mechanical Engineering, K.J College of Engineering & Management Research, Pune, Maharashtra, India*

[3] *Vaftsy CAE, India*

---***---

Abstract - *Robotic structures are challenging because of the involving of dynamic forces. These dynamic forces further amplify themselves during emergency stop operation. Further a pick and place operation has its own operating frequency, if this frequency resonates with the structure it results in dramatic failure so a structure that supports such an operation needs to be stable both in static condition as well as in dynamic condition. The frequency analysis of the outer structure depends on the load by the pedestal and the robot which is totally mounted at the center. The main aim is to avoid the resonance occurrence between the structure with the robot and the conveyor. The software used for analyzing the frequency is ANSYS.*

Key Words: **FEA, Vibrations, Optimization, Structure Design, Natural frequency, Resonance.**

1. INTRODUCTION

The techniques of analysis and simulation of mechanical systems using the finite element method allows researchers and mechanical engineers to build mathematical models and to analyse the static and dynamic behaviour of the structural elements directly on the computer, and optimization calculations, simulations, studies of similarity, etc.[1]

Fig -1: Similar Structure to be designed

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]

2. LITERATURE REVIEW

Irjet Zhijun Wu et al has studied two 3D finite element models of the container crane by ANSYS respectively based on two common doorframe structures of compound type and single brace type. And then stress, deformation, mode shapes are analyzed and compared. As dominant heavy lifting appliances in large ports, container cranes are becoming larger and heavier. Main steel structure of the crane takes up one third of overall cost and therefore there is a great significance on lightweight design. [2]

Jaydeep Roy et al has studied Structural Design Optimization and Comparative Analysis of a New High-Performance Robot Arm via Finite Element Analysis. Design objectives for the new arm include large (1-2 meter) workspace, low weight, 5 kg payload capacity, high stiffness, high structural vibration frequencies, precise joint-level torque control, a total of three degrees-of-freedom, and mechanical simplicity.[3]

Xiaoping Liao et al have studied a modal analysis method of the base of welding robot by using the finite element analysis software ANSYS release10.0. The base suffers large dynamic stress as shock and vibration while welding robot works, which on the one hand lead to lower precision of welding, on the other hand, may cause the base damaged. Thus dynamic analysis is necessary for design of the base of welding robot. [4]

3. PROBLEM STATEMENT

The brief objectives of the project are

1. Modal analysis and optimization of structure designed by company as per required results.

2. Optimizing the natural frequency of the structure using FEA.
3. Harmonic response analysis for validation with experimental setup.

4. GIVEN MODEL& MODAL ANALYSIS

CASE -1: Sample Table format

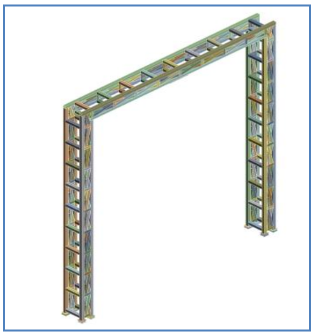

Fig -2: Geometry which was provided after the trial error design

Optimized For CASE – 01, the geometry was provided by the company after the trial error design. The geometry consists of hollow beams with dimensions 88.9mm X 88.9mm X 4.5mm. The material for complete structure is structural steel. Boundary Conditions:

1. Forces applied as per the data of pedestal.
2. Moment calculated on pedestal and applied on the top plate.
3. Fix support at the end of bolts to be mounted on the bed.
4. Frictional support between base plate and the bed.

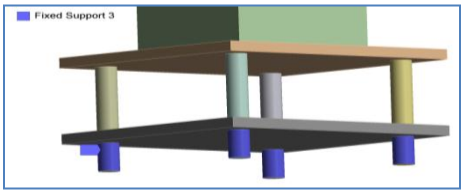

Fig -3: Fixed support at the base bolt

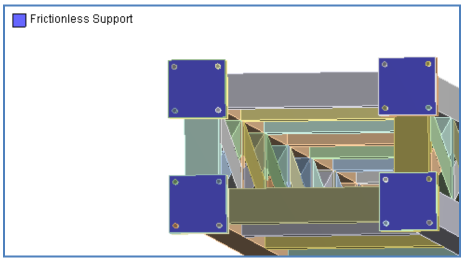

Fig -4: Frictionless support at the base plate

The fixed support on each bolted member is shown in fig.4.

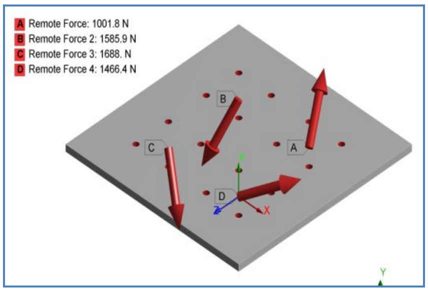

Fig -5: Forces on the plate due to pedestal

The forces created due to pedestal on the plate are shown. On the set of 4 holes a total remote force is applied whch is shown in fig.5

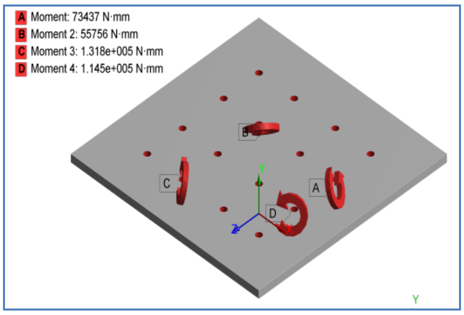

Fig -6: Moment on the plate due to pedestal

Also the moment is created due the pedestal. The moment hence applied is shown in fig.6.

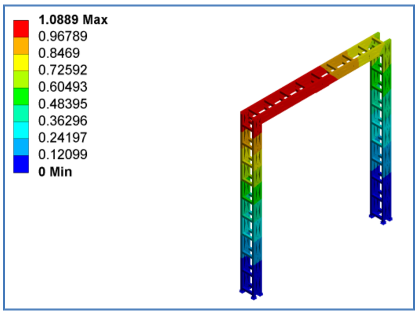

Fig -7: Deformation in the structure of CASE – 01

The deformation of the structure in CASE – 01 is 1.0889 mm. The red area indicates maximum deformation due to the forces and self-weight of structure.

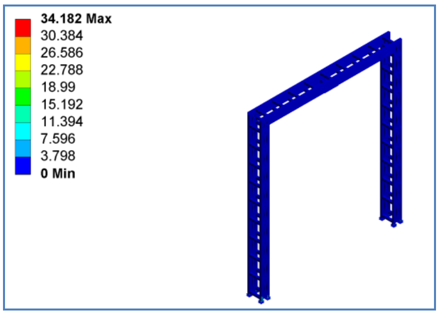

Fig -8: Stresses in the structure of CASE – 01

For taller and longer beams the stress analysis is done only to find the stress concentration area. Even after reducing the stresses to 34.182 MPa, addition of the material is continued because the aim is to increase the frequency above 10 Hz.

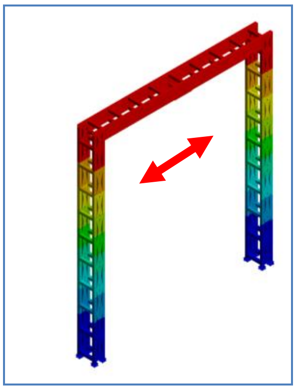

Fig -9: Modal analysis for CASE – 01

The modal analysis was carried on the structure. To increase the frequency we added the horizontal members on the sides of the structure which results frequency of the structure is 4.9884 Hz.

Similarly, analysis on more 7 cases with changes in geometry were carried out and the results were tabulated as

Table -1: Results for all cases

Preparation of Manuscript			
No. of CASES	**Maximum Deformation mm**	**Maximum Stress MPa**	**Frequency Hz**
1	1.0889	34.182	4.9884
2	1.084	38.795	5.3653
3	1.0457	39.89	5.3483
4	0.20029	13.794	7.973
5	0.19647	13.686	9.7477
6	0.1906	12.304	9.177
7	0.18861	12.038	10.308
8	**0.17838**	**9.8731**	**14.448**

5. HARMONIC RESPONSE

The harmonic response analysis was carried on the final case. The geometry of last case is shown in fig.6.

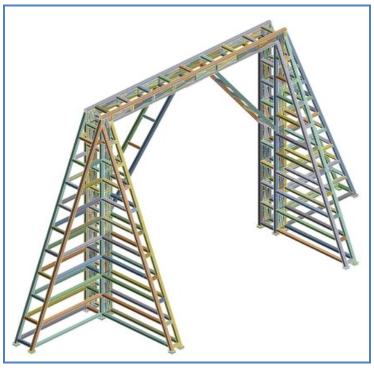

Fig -10: Geometry on which harmonic analysis was carried out

The analysis settings changed for harmonic response are as follows:

Options

Range Minimum	13 Hz
Range Maximum	15 Hz
Solution Intervals	20

Harmonic response for range of frequency:

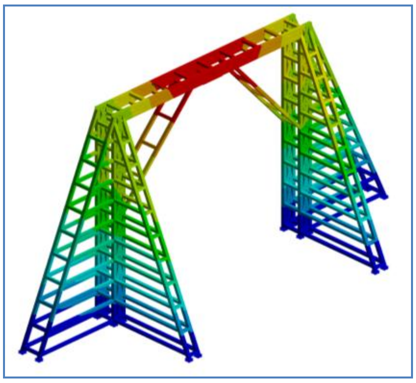

Fig -11: Deformation in Harmonic response

The frequency range for harmonic response is from 13 Hz to 15 Hz. The solution intervals considered for this is 20. The maximum deformation for the response is 4.017 mm for frequency 14.36 Hz. This deformation is calculated on ANSYS for the structure. The fig.11 shows the direction in which the structure deflects due to applied frequency.

6. VALIDATION & DISCUSSION

Validation input given by company:

Specifications for robot arm:

Useful load: 6 kg

Max. Reach: 901 mm

Number of axes: 6

Repeatability: <± 0.03 mm

Weight: 52 kg

Floor mounting positions

Compact control: KR C4

Degree of protection: IP 54

Analysis:

Measuring sensor used & its specification

Model: LK-G5001/LK-H008W

Interface RS – 232C: Baud Rate: 9600 to 115200 bps

Reference Distance: 8 mm

Measuring Range: ±0.5 mm

Source of light: Red semiconductor laser, 655nm, 0.3mW

Experimental Results			Result obtained on analysis software
No. Of Tests	Max. Deformation mm	Average mm	Max. Deformation mm
01	4.6	4.67	4.017
02	4.75		
03	4.65		

From the table, the percentage error is calculated which is 13%.

7. CONCLUSIONS

Hence, the paper concludes that the analysis done on software gives approximate results and the error is also calculated in order to prove corrections in the analysis carried out.

8. PROPOSED WORK

1. Optimizing the geometry of the structure.
2. Optimization of weight can be done as this geometry was finalized according to availability of material.
3. Validation of results with more accurate results can be obtained by carrying out more number of analysis on trial structure.

ACKNOWLEDGEMENT

It gives us great pleasure to submit this paper on "Finite Element Analysis of Pick and Place Robot Structure". We take this opportunity to thank Prof. C. M. Gajare for his valuable guidance and his deep interest throughout the study of this paper.

REFERENCES

[1]. Adrian GHIORGHE, "OPTIMIZATION DESIGN FOR THE STRUCTURE OF ANRRR TYPE INDUSTRIAL ROBOT", U.P.B. Sci. Bull., Series D, Vol. 72, Iss. 4, 2010, ISSN 1454-2358

[2]. Zhijun Wu, Kailiang Lu, HuiqingQiu, "FINITE ELEMENT COMPARATIVE ANALYSIS OF TWO DOORFRAME STRUCTURES IN CONTAINER CRANE" College of Mechanic Engineering, Tongji University, Shanghai, 201804, P.R.C

[3]. Jaydeep Roy, Randal Goldberg, and Louis L. Whitcomb, "STRUCTURAL DESIGN AND ANALYSIS OF A NEW SEMI-DIRECT DRIVE ROBOT ARM: THEORY AND EXPERIMENT" Department of Mechanical Engineering Johns Hopkins University

[4]. Xiaoping Liaol, Changliang Gong, Yizhong Lin, Weidong Wang, "THE FINITE ELEMENT MODAL ANALYSIS OF THE BASE OF WELDING ROBOT" College of mechanical & engineering, Guangxi University, Nanning, Guangxi, 530004, China