Journal of the Maharaja Sayajirao University of Baroda
ISSN: 0025-0422

ROBOT PEDESTAL DESIGN AND ANALYSIS

Sripal.V[1], Dr.P. Santosh Kumar Patra[2]
[1]Assistant Professor, St.Martins Engineering College, Secunderabad, Telangana
[2]Professor&Principal, Department of computer science and engineering, St. Martin's Engineering College, Secunderabad

Abstract - *Todays automation scenario robot is very essential part. There is many standard robot manufacturers which manufactured the standard sizes of the robot. According to industrial application to achieve desired position of the work robot required its customized height of the robot arm.to achieve desired height robot requires its own pedestal.so initially we design the cross section beam for the robot pedestal with considering different parameters of the loads and developing the CAD model for the according to the requirement. For modeling and analysis we use the ANSYS 15.0 software. We apply boundary condition on that model and doing FEA of that pedestal according to analysis result we change the body structure. We find the total deformation and equivalent stress to safe that model.*
KEYWORDS: *Robot pedestal,Ansys,Stress, Defoemation*

1.INTRODUCTION

Robot is automatically operated reprogrammable machine which used to reduce the problem human labor. Robots can do industrial work with great accuracy and continually. Now days for mass production in industries we are using robots, which produce very precious part with great speed. According to the height of robot required we design the pedestal for that. We are analyzing the which type of failure occurs in the pedestal structure, at which load and moment,. After that firstly we make CAD model of robot pedestal using FEA software i.e. ANSYS 15.0.Applying boundary conditions such like forces on robot pedestal and moments applies on the robot pedestal. After model analysis we will find, von- misses stresses and total deformation of robot pedestal. After result analysis of stress, deformation and mode shape we will decide where the pedestal is safe for given loads as well as moments. We are designing robot pedestal such a way that pedestal having increasable stress and load bearing capacity.

Fig1: Robot Pedestal

2.LITERATURE REVIEW

[1] Xiaoping Liao et al (2010)In this article, the analysis software ANSYS release 10.0 have been used in the modal analysis of the base of welding robot, and the natural frequencies and mode shapes of the first ten orders have been computed quickly and directly.

[2] Gwang-Jo Chung et al (2010) they obtained the maximum reaction force for each joint that could be used for static rigidity analysis. Next, through the mode analysis, theyestimated the natural frequency for the overall assembled structure and compared it with the experimental result to identify the accuracy and the reliability of the FEM models.

[3] Zhijun Wu et al(2011) In this paper, they build two according 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.

[4] Jeevan et al(2015)studies the modeling and analysis of robot arm using ansys . The mechanical design, structural analysis, and results verification of a new high performance semi-direct drive robot arm. A design optimization methodology employing finite element analysis (FEA) is reviewed, and a resulting arm design is reported.

3.MODAL ANALYSIS OF THE ROBOT PEDESTAL

3.1 modeling and meshing

In the majority most of the cases of the designconsideration structural steel is used as a material. Hollow rectangular tubes of structural steel are used for the pedestal the supportive members are also hollow tubes at the centerof pedestal. At bottom end four plates are attached to the column as per shown in the fig2. For mounting on the foundation robot pedestal bolts are used. The pipes are attached to each other by the welding process. Material properties are shown following table.

Table 1 Material properties of pedestal

Material	Structural steel
Modulus of elasticity	190 GPA
Yield strength	355Mpa
Tensile strength	470Mpa
Density	7850kg/m3

The pedestal for the Welding robot is modeled in ANSYS15.0 package of modeling and meshing as shown in Fig.2 . In this meshing we use the 20 mm size to model the pedestal for each part of robot pedestal and the material property of the pedestal has been defined in the main menu of ANSYS15.0, which is made of structural steel with a modulus of elastic of "190Gpa" and density of "7850kg/m3". After the meshing we get the 150570 nodes and 27892 elements and were generated at element size 20 mm.

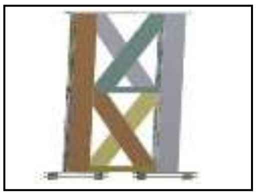


Fig.2Model of pedestal

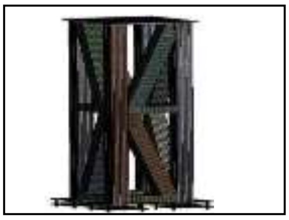


Fig.3 Meshing

3.1.Boundary condition

After model analysis we find of six modes of natural frequency the figures of natural frequency with six modes as per given below. In table shows below natural frequency ateach mode shape.

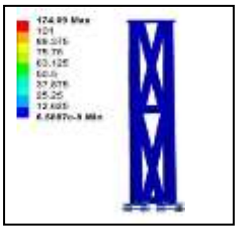
Table 2 Natural Frequency Of Pedestal

Sr no	Frequency
1	7.2793
2	10.741
3	19.053
4	29.172
5	85.355
6	130.43

After completing the model of the pedestal in the workbench of the ANSYS we are going to meshing the model by using the Hex dominant and sweep method for the meshing. Boundary condition are applied on the pedestal are vertical force Fy=1362N and Horizontal force Fz= -1297N. Also Tilting moments are Mx=1152Nmm and Mz=880Nmmand fix supports are provided at bottom circular surfaces of bolts below the surface plate and With addition of frictionless support to the plates contact with that bolts.

3.2 Viewing the results

After applying the forces analysis is done and we get the following total deformation and Equivalent stress figure. The maximum stress is 146 N/mm^2 which is below the allowable stress and total deformation is 1.75 mm which is also safe level by using the L/1000 mm criteria.



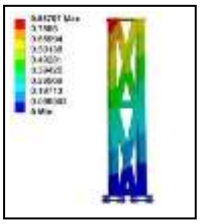
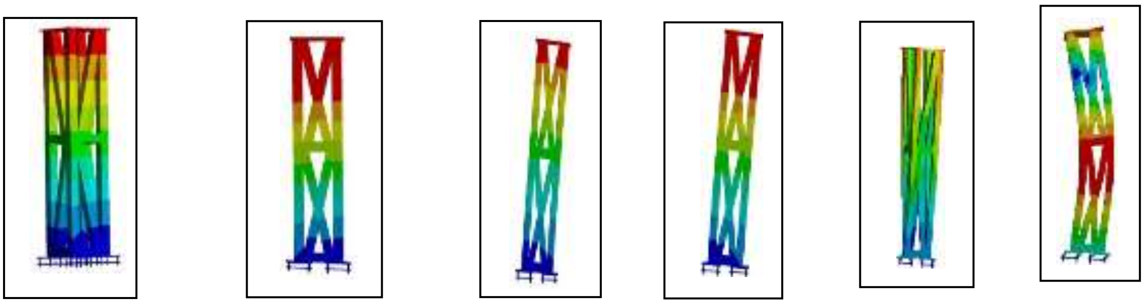


Fig 4 equivalent stress fig 5 total deformation

After model analysis we find of six modes of natural frequency the figures of natural frequency with six modes as per given below. In table shows below natural frequency ateach mode shape.

Sr no	Frequency
1	7.2793
2	10.741
3	19.053
4	29.172
5	85.355
6	130.43

Table 2 Natural Frequency Of Pedestal



(a) (b) (c) (d) (e) (f)

Fig.6 (a) 1st mode shape (b) 2nd mode shape
(c) 3rd mode shape (d) 4th mode shape
(d) 3rd mode shape (f) 4th mode shape

4.CONCLUSIONS

The finite element approach presented in this paper was implemented by the pack-age ANSYS 15.0. we obtained the equivalent stress and total deformation for pedestal also we found the the natural frequencies and the corresponding mode shapes of the first six orders of the base obtained by modal analysis obtained after applying boundary condition by the finite element model, and the conclusion are as follows: Natural frequency of the pedestal is minimum at first mode and maximum at sixth mode it increases from 7 Hz to 130 Hz.

5.REFERENCES:

[1] *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*

[2] *Gwang-jo chung, doo-hyung kim,"structural analysis of 600kgf heavy duty handling robot"2010 principal research engineer in robotics lab. korea institute of machinery & materials daejon, korea.*

[3] *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*

[4] *Jeevan, dr. amar nageswara rao, "modeling and analysis of robot arm using ansys"ijsetr issn 2319-8885 vol.04,issue.33, august-2015, pages:6692-6697*