Design Analysis and Optimization of Robot Pedestal

G.A.Yadav, A.M.Pirjade, M.M.Jadhav,Vinaay Patil

***Abstract*— A robot is any automatically operated machine which replaces and reduces human effort, though it may not look much like a human being or function in a humanlike manner. Now days in industries robots having very important role, from material handling to quality control we are dependent on robots. In this project we are analyze the robot pedestal. Pedestal means the main supportive member which is the base of the robot and it gives support as well as stabilization to the robot. By using ANSYS 15.0 in workbench we firstly make 3D geometry of robot pedestal as per dimension. After those applying boundary conditions like weight of robot, forces applied on its body as well as moments on the pedestal. After that analyze stresses, von-misses stress, total deformation and the natural frequency of robot pedestal. Finally after getting results we will decide the robot pedestal design is safe or not.**

***Index Terms*— Finite element analysis, modeling, optimization, robot pedestal.**

I. INTRODUCTION

Robot is automatically operated machine which work like human being or used to reduce efforts of human being. Robots can do household work as well as industrial work with great accuracy. Now days for mass production in industries we are using robots, robotic work is accurate as well as reliable because of that we are using maximum amount robots for mass production.

When there continuous use of robots and their parts then there is chance to failure occurs in robot or there is damage in robotic parts after some years. There is need to design the robotic part newly, analyze such like that there efficiency is greater than that of the old parts and stress or load bearing capacity is more than the previous one. Which results in increasing life of robots and there is profit to industries.

While using continuous robot there is damage is robotic parts or because of continuous vibration there is failure occurs in pedestal. We are analyzing the failure in pedestal. Which type of failure occurs, at which load, after how many revolutions of robot. After that firstly we make geometry of robot pedestal using FEA software i.e. ANSYS 15.0.Appling boundary conditions such like load on robot pedestal and moments applies on the robot pedestal. After model analysis we will find out natural frequency, von-misses stresses and total deformation of robot pedestal. Finally after result analysis 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. And having capacity to work at high speed.

G.A.Yadav, P.G. Student, Department Of Mechancal Engineering, A.D.C.E.T. Ashta, Dist. Sangli, Maharashtra.

A.M.Pirjade, Assistant Professor, A.D.C.E.T. Ashta, Dist. Sangli, Maharashtra.

M.M.Jadhav, Assistant Professor, A.D.C.E.T. Ashta, Dist. Sangli, Maharashtra.

Vinaay Patil, Vaftsy Cae, Pune, India.

From figure given below we can see the location or the structure of robot pedestal.

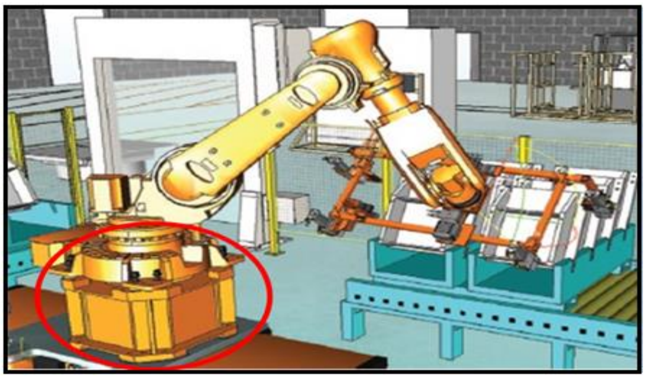


Fig. 1: Robot Pedestal

II. LITERATURE REVIEW:

X. Liao (2010) in this project work they are work on the base of the welding robot. By using ANSYS-10.0,they are find out the natural frequencies, mode shapes of base of welding robot & dynamic analysis of robot. There results from model analysis are , the upper edge & tail edge of the base have a larger vibration which are susceptible to be fatigued & damaged. They found maximum amplitude at the 7th natural frequency was 39.249 Hz. By using ANSYS -10.0 software the results are quick & reliable

G. Chung et al. (2010) this work is based on design of heavy duty industrial robot .After that they tested or find out the static & dynamic analysis for heavy duty applications i.e. at 600kgf(max)weight. Finally they compare the experimental as well as FEM results. They are analyzed that the elastic rigid body dynamics to estimate the robustness of the robots parts during working motion of robot. The results are found that frequency of reaction forces from multi body analysis is 6.67Hz & Natural Frequency was 18 Hz.

J. Roy et al. (1997) they are working on structural design of a new high performance robot arm. They are making the comparative analysis for four different two degree of freedom link edge candidates by using FEM. The results are found by using band drive mechanism are having overall best performance. The practically useful workspace with toggle point singularities is severally restricted due to poor geometry near the toggle points.

L. L. Whitcomb et al. (1999) this paper makes comparative structural analysis of 2-degree of freedom semi direct drive linkages of arms. This four optimized linkages are (1) parallelogram (2) double parallelogram (3) double kite (4) band drive. By making comparison of these linkages, they found that there was significant improvement in stiffness & fundamental frequency & weight optimized by iterative FEA design optimization.

R. P. Goldberg et al. (2004) they reports the design, analysis & comparative results of structural & FEA analysis of semi-direct drive robot arm. They found the experimentally lowest natural frequency of designed arm is 66 Hz in the 90o configuration, but frequencies in all other orientations are higher. Also they are found that peak tip velocity of arm is greater than 6m/s & the accelerations are greater than 7g with maximum position tracking errors less than 5%.

M.H.F.Dado et al. (2000) in this study they work on dynamic response of a planer 2-R robot with flexible joints is investigated. They are modeled servo stiffness & material damping of the drive system. This study concludes that servo damping plays an important role in dynamic behaviour of system

A.H.Soni et al. (1987) they study analyzed the surgery instruments. Firstly they will present the design data of surgical instruments after that they study on the applications of the instrument to collect data describing the vertical alignment of a scoliotic spine in three dimensions, before & after surgical correction. Lastly they compare the relative efficiency by using various surgical techniques involving Harrington rod, Lugun rod & Dwyer apparatus etc.

W. Shijun et al. (2002) they work on the optimal design for robot structure. Optimal design id not automatically modify it need to modify automatically & computed repeatedly. For that purpose they used APDL language of ANSYS software to generate an optical control program which is used to make optimal procedure to run automatically & it will improve the optimal efficiency.

J.H.Varma et al. (2015) this project work on the robot gun support structure for welding purpose of light vehicle door frame. Firstly they design & analyze welding subsystem with their functionality & applications. They design a structure of welding gun support which is able to move to multiple locations quickly even causing force in tunes of 1.5 times of gravity.

III.MODAL ANALYSIS OF THE ROBOT PEDESTAL

A. modeling and meshing

In the majority of the base of the robot pedestal, structural steel is the principal material used. The four hollow square tube columns with square plate on its top end of the column hollow tubes. Also the supportive members are hollow tubes at the centre, at the base and at inclined positions. At bottom end of the hollow column tubes there are four foundation bolts with two supportive pates at each hollow column tube, is the main geometry of robot pedestal. The material properties of the base of robot pedestal are as shown in Table l.

Table1: MATERIAL PROPERTIES OF THE BASE.

Material	Structural steel
Density	7850kg/m^3
Modulus of elasticity	190Gpa
Yield strength	355Mpa
Tensile strength	470Mpa

The base of the Welding robot is modeled in ANSYS release 15.0 of modeling and meshing as shown in Fig.2 and Fig.3. In this analyze, element is used 13 mm to model the pedestal for each element of robot pedestal. The material property of the robot pedestal has been defined in the main menu of ANSYS15.0, which is made of structural steel with a modulus of elastic of “190Gpa” a density of “7850kg/m^3”. After the meshing was done there are 27891 elements and 150568 nodes were generated at element size 13mm.

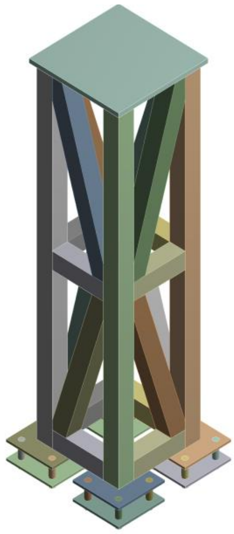


Fig.2. Creating a model

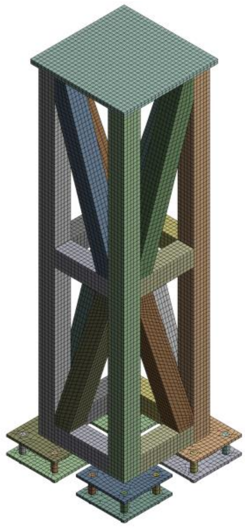


Fig.3. Meshing the model

B. Loading and Solving

Up to this stage the geometry and done meshing is done with hex dominant method. After this meshing applying forces as well as moments on the pedestal. The forces applied on pedestal are Fy=1362N, Fz= -1297N and moments are Mx=1152Nmm, Mz=880Nmm.The supports are provided at bottom circular surfaces of bolts below the surface plate. With addition of frictionless support to the plates contact with that bolts.

C. Viewing the results of the operation

When the analysis was done we will find out results of total deformation and the equivalent stress. From figures given below in figure 4 we will see total deformation and in figure 5 we will see equivalent stress of robot pedestal.

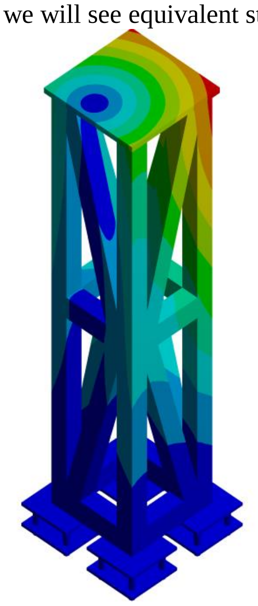


Fig.4. total deformation

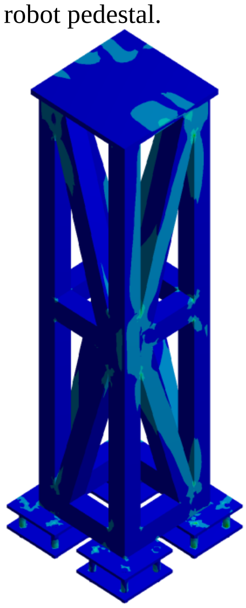


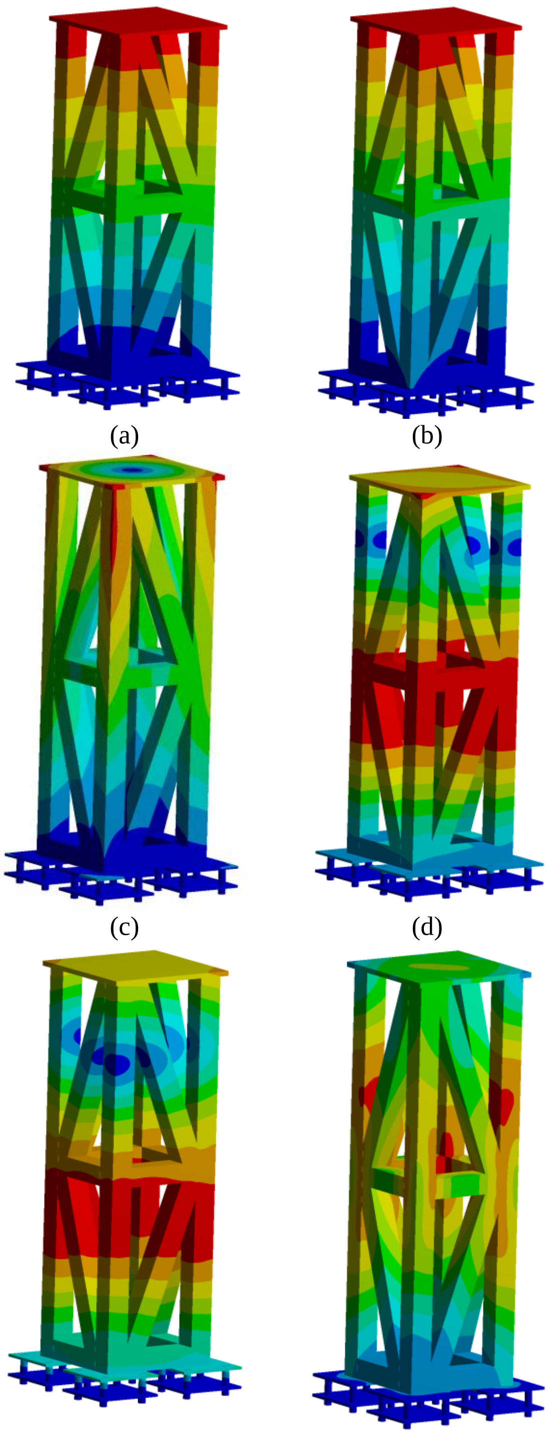
Fig.5.equivalent stress

Also after model analysis we will find of six modes of natural frequency. The figures of natural frequency with six modes as per given below. In table below we will see natural frequency at each mode.

TABLE 2: THE NATURAL FREQUENCY OF FIRST TEN ORDERS

Mode	Natural frequency/Hz
1.	69.536

2.	77.485
3.	206.63
4.	330.42
5.	435.15
6.	446.36



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

Fig.6. (a) 1st mode shape (b) 2nd mode shape
(c) 3rd mode shape (d) 4th mode shape
(e) 5th mode shape (f) 6th mode shape

III. CONCLUSION:

The finite element approach presented in this paper was implemented by the general-purpose finite element pack-age ANSYS release 15.0. The natural frequencies and the corresponding mode shapes of the first six orders of the base obtained by modal analysis obtained after the modeling, meshing, loading for the finite element model, and the conclusion are as follows:

1) Results from the modal analysis showed that, the Natural frequency of the pedestal is minimum at first mode and maximum at sixth mode it increases from 69.536Hz to 446.36 Hz.

2) The result shows that the total deformation at the loaded condition was 0.17539mm and equivalent stress was 32.472 Mpa.

3)At the sixth mode we are getting maximum frequency i.e. 446.36Hz and getting maximum deformation as well as equivalent stress, at this position the structure of robot pedestal is safe it is not get damaged at maximum load condition.

4) The correct analysis of structural characteristics and accurate constraints and loads to the model play an extremely important role in to obtain a reasonable result. In modal analysis of the base, we especially identify equivalent unit that make sure the result is direct and readable.

5) The results obtained are showing that the capabilities of the ANSYS software can be used successfully as a and reliable tool for prediction of the vibration behavior of the robot pedestal. The modeling method presented in this paper can be employed for the design of the pedestal.

REFERENCES:

[1] The Finite Element Modal Analysis of the Base of Welding Robot-Xiaoping Liao, Changliang Gong; 2010 3rd International Conference on Advanced Computer Theory and Engineering (ICACTE), 2010 IEEE.

[2] Structural Analysis of 600Kgf Heavy Duty Handling Robot Gwang-Jo Chung, Doo Hyung Kim; Principal Research engineer in Robotics Lab. Korea Institute of Machinery & materials Daejon, Korea, 2010 IEEE.

[3] Structural Design Optimization and Comparative Analysis of a New High-Performance Robot Arm via Finite Element Analysis-Jaydeep Roy, Louis L. Whitcomb; proceedings of the 1997 IEEE,International Conference on Robotics and Automation Albuquerque, New Mexico - April 1997

[4] Comparative Structural Analysis of 2-DOF Semi-Direct-Drive Linkages for Robot ArmsJaydeep Roy, Louis L. Whitcomb; IEEE/ASME Transactions On Mechatronics, Vol. 4, No. 1, March 1999

[5] Structural Design, Analysis, and Performance Evaluation of a New Semi-Direct DriveRobot Arm: Theory and Experiment- Jaydeep Roy, Member, IEEE, Randal P. Goldberg, and Louis L. Whitcomb, Senior Member, IEEE IEEE/ASME Transactions On Mechatronics, Vol. 9, No. 1, March 2004.

[6] A.H.Soni and M.H.F. Dado, Dynamic response analysis of 2-R robot with flexible joints. IEEE Conf. on Robotic and Automation (1987)

[7]FEM Optimization for Robot Structure-Wang shijun, zhao jinjhan; IEEE ICIT'12,Bangkok, Thailand, 2002 IEEE.

[8] Analysis and Optimization of Robot Gun Support Structure for Welding of Light Weight Vehicle Door Frame - Jatin H. Varma, Onkar G. Sonare, ISSN (Print) : 2321-5747, Volume-3, Issue-6,2015

[9]Application of passive robot in spine surgery, Soni, A.H. Oklahoma State university, Stillwater, OK. Gudavalli, M. ; Herndon, W. ; Sullivan, J. Robotics and Automation. Proceedings. 1987 IEEE International Conference on (Volume:4)