International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 04 Issue: 07 | July -2017 www.irjet.net p-ISSN: 2395-0072

# FINITE ELEMENT BASED ANALYSIS OF ROTATING ROBOT PEDESTAL

**Mr. Pramod Sarode[1], Prof. S. M. Magar[2], Mr. Vinaay Patil[3]**

*[1] Scholar, ME Design, Department of Mechanical Engineering, D.V.V.P.COE, Ahmednagar, Maharashtra, India*
*[2] Assistant Professor, Department of Mechanical Engineering, D.V.V.P.COE, Ahmednagar, Maharashtra, India*
*[3]Vaftsy CAE India*

---------------------------------------------------------------------***---------------------------------------------------------------------

**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 working in a human like manner. Most robots are designed for specific functions within a custom environment for performing aerial tasks. Robots require its own custom manufactured pedestal structure i.e base structure, custom built to size and strength in order to ensure immobility while firmly supporting the robot.in this project we analyze the pedestal which supports the two robots. Firstly we decide the model and created on the ANSYS 15.0 workbench. Then meshing the model in ansys with suitable element and apply the boundary conditions like as forces, moments, fix support. Then getting the result of total deformation and equivalent stresses we decide the design is safe the model of first pedestal and then find the force reaction of the second pedestal with that values safe the final model. The procedure and result of the analysis are described in this paper*

***Key Words*: *robot pedestal, finite element analysis, optimization, equivalent stress, total deformation***

## 1. INTRODUCTION

An industrial robot is a manipulator designed to move materials, parts and tools, and perform a variety of programmed tasks in manufacturing and production settings. The industrial robot is a good for many applications. It is most often used for arc welding, material handling, and assembly applications. They are grouped according to number of axes, structure type, size of work envelope, payload capability, and speed. Robots require specialized supporting structure to accurately hold the workpiece during the operations. Most robots are designed for specific functions within a custom environment for performing aerial tasks. Each robot usually requires its own custom manufactured robot pedestal, custom built to size and strength in order to ensure immobility while firmly supporting the robot. Each robot usually requires its own custom manufactured Robot Pedestal, custom built to size and strength in order to ensure immobility while firmly supporting the robot. Most robot suppliers do not offer a Robot Pedestal as per client application. Due to the nature of the custom environment and application, no standard Robot Pedestal design will work every time. In fact, a custom height Robot Pedestal will be required 99% of the time.it is crucial part of design when two rotating pedestal are attached to the one pedestal it should be sustain the dynamic and static load within the operation so failure cannot obtained. Fig 1 shows that the arrangement of the pedestal for rotating robot. It shoes that tentative model of the pedestal.by using the FEM we safe that structure with the boundary condition given by respective catalogue.

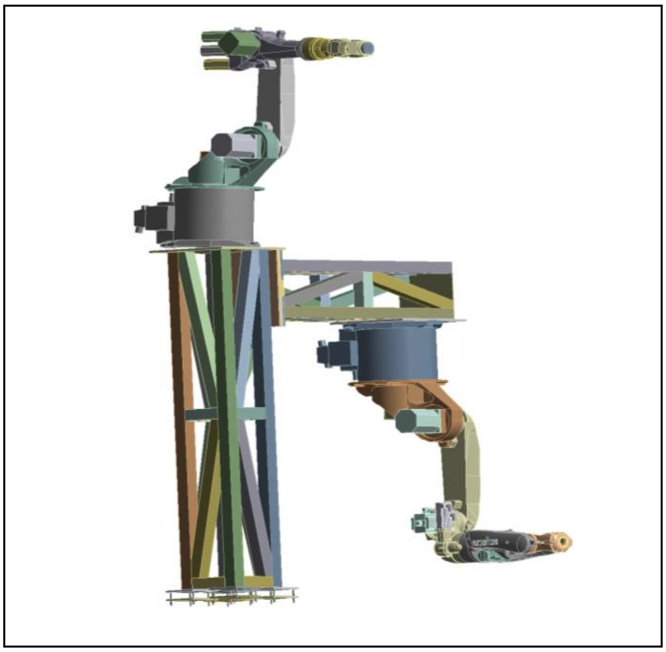


**Fig.1** Similar Structure to be designed

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 analyze the static and dynamic behavior of the structural elements directly on the computer, and optimization calculations, simulations, etc. Approximate numerical solutions, obtained through modeling for the proposed problems, have the advantages such as it can be applied to bodies and real phenomena, regardless of their degree of complexity are converge to solutions of proposed problems (results may be obtained with the desired accuracy) also you can view the pictures, charts, graphs - intuitive and more diverse than in the case of exact solutions. allows to obtain a solution in a reasonable time with the economically advantage.[1]

## 2 METHODOLOGIES

First cross-section of the beam is selected for the pedestal model by using the boundary condition values which are used for calculation. By using that beam Model is prepared in ANSYS Workbench 15.0. After that it is ready to meshing in ANSYS workbench15.5.find the appropriate method of element for meshing. Fine meshing is done by the node optimization, after those proper boundary conditions are applied. Final results are obtained after solution Conversions takes place. In following figure these steps are shown in the flowchart.

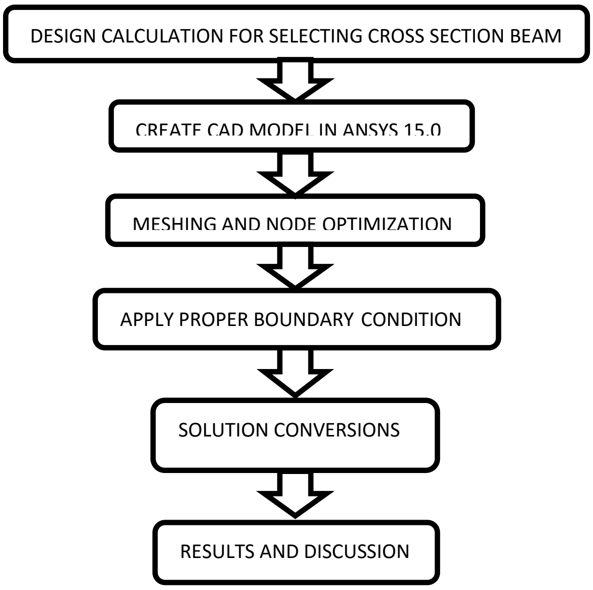



**Fig2.** Flowchart of methodology

## 3 PROBLEM STATEMENTS

Robot Pedestals are the foundations which take the load of the entire robot assembly, the robot, the gripper and the panel that is being manipulated. Hence the sums of total of the dynamic and static loads come on the pedestal, which makes it crucial for the stability of the system

In this project a further challenge is added to this mix, in which the pedestal rotates by 90 degree and moves a certain distance to make way for another robot in the same vicinity. This is challenging as the base of the pedestal will now have to handle this rotation movement and incorporate it in its design.

### 3.1 Objective

The objective of the project is analyze the effect of the additional motion have on the stability of the system

- The idea is to provide an optimized solution
- Optimization parameters will be material and dimensions of the pedestal
- It will have to conform to life standards set by the client

## 4. FINITE ELEMENT ANALYSIS

The Finite Element analysis rotating robot pedestal is to be done on ANSYS Workbench 15.0. The purpose of analysis was the equivalent stress and total deformation observed in the in the structure within allowable stress.by analyzing the first pedestal the find the reaction forces for second pedestal and apply this forces on the first pedestal and safe the design of structure given allowances.

### 4.1 Geometry

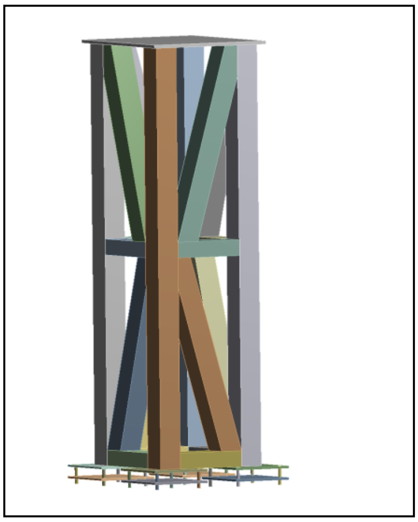


**Fig.3** robot pedestal model

Geometry consists of vertical and horizontal hollow rectangular cross sectional beam which is 75.0 X 75.0 X 4.0 . The cross section beam is selected as standard size from IS-4923 chart with the reference of combine loading calculation. Upper side having the base plate for mounting the robot as per dimensions of the base of the robot. Inclined member also add for giving strength and can sustain the load of second pedestal.

The material properties for analysis are shown in table

**Table -1:** material properties

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

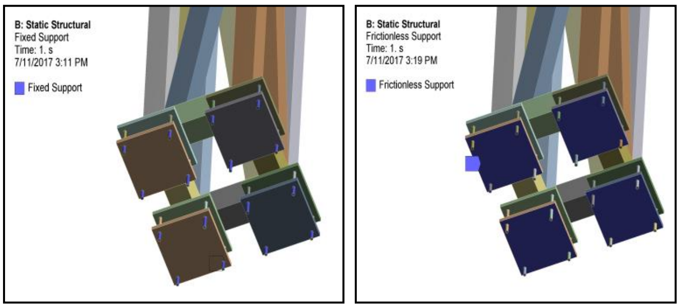
### 4.2 Meshing of model

Finite Element Method analysis tool reduces degrees of freedom from Infinite to Finite with the help of meshing. There are several methods for the meshing of model. As per application different methods are used. In this problem we have to structural analysis so Quad, hex elements are preferred over trias, tetras and pentas.so we use the sweep method for all parts except inclined cross beam. For inclined cross member beam use hex dominant method for meshing.

### 4.3 Boundary conditions:

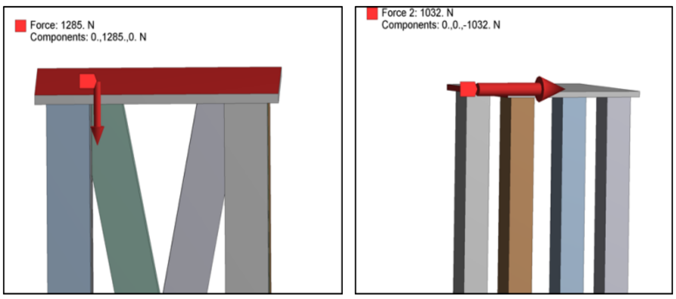
Following are the boundary condition used for analysis:
1. Forces and moment applied as per the data of kuka robot catalogue.
2. Fixed support at the bolt support of pedestal.
3. Frictionless support at the base plate of pedestal

The figures given below shows the applied locations for each boundary condition.




(a) (b)

**Fig.4**. Base plate boundary condition




(a) (b)

**Fig.5**Boundary condition for force and moment on the pedestal

Fig.4 shows the boundary conditions, in which fig. (a) Shows the fixed support applied on the base plate bolt. These are bolted into the base to provide holding support to the complete structure.in fig.(b)shows that the base plate are applied with the condition of frictionless support. The contact between the foundation and these plate faces is frictionless. Fig.5 shows the forces and moment on the robot mounting plate.in fig. (a), the vertical force due to robot which is equal to 1285N. These forces are applied on the faces of mounting plate in downward direction only. It shows also the horizontal force 1032N which is applied on the side of robot mounting plate which is inward or outward in direction. Fig.5 (b) shows the moments on the robot mounting plate.in this fig shows the tilting moment due to robot. This moment are applied on the faces of mounting plate in anticlockwise or clockwise direction which is 595Nm. These moments are applied on the upper face of mounting plate. It also shows that torque 741Nm on the face which is applied on the clockwise direction only. Because the result is not changes with the changing direction of moment.

### 4.4Node optimization:

FEA method gives the approximate values of solution. When we decrease the element size from course to fine nodes results are varied with some difference of percentage.so we use the fine mesh for the analysis.to find the fine element size we are going to node optimization for further analysis.

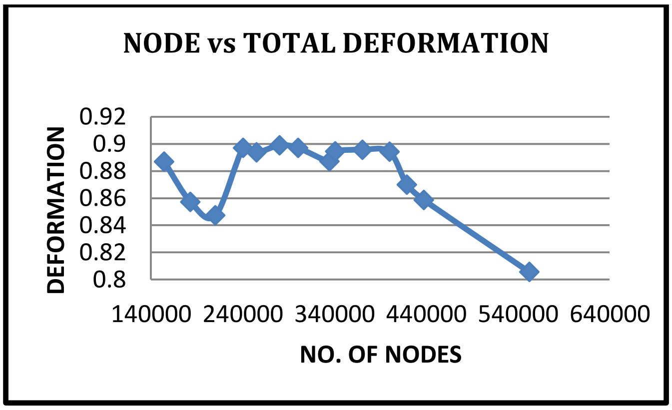



**Fig.6** graph of node vs deformation

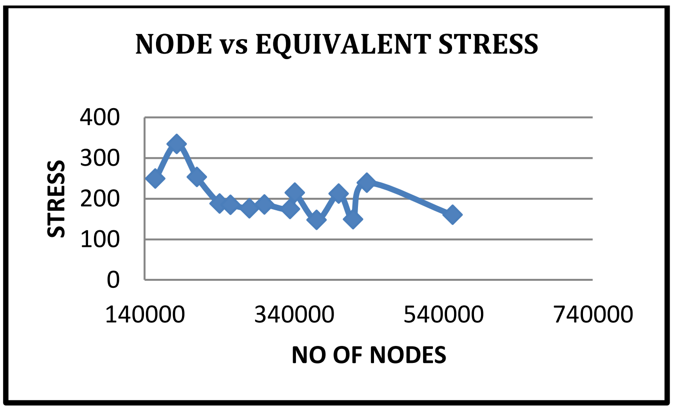



**Fig.7** graph of nodes vs stress

By observation of the both graphs we choose the value of no of node is around the 437000 .This is the fine mesh size for the further analysis of robot pedestal.

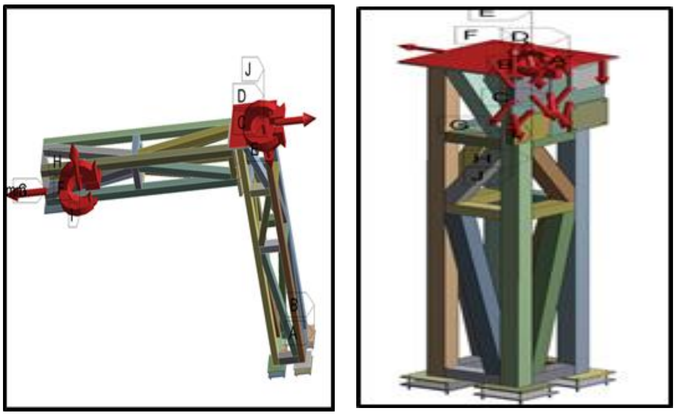
### 4.5 Plate thickness optimization:

**Table 2** result of plate thickness optimization

| Sr no | Plate Size mm | Equivalent Stress N/mm² | Total deformation mm |
|---|---|---|---|
| 1 | 10 | 238.81 | 0.85086 |
| 2 | 11.5 | 125.23 | 0.75316 |
| 3 | 11 | 143.63 | 0.7732 |
| 4 | 12 | 115.91 | 0.72123 |

At optimized node size 437000 the maximum equivalent stress is 213Mpa safe this model it should be below the 150Mpa which is allowable stress. We find the stress concentration area which is find by the probe in the ansys.by observing the stress concentration found that maximum stress is produced on base plate of pedestal.by increasing material size i.e. width of the plate from 10 mm to 12 mm we get the optimum value for plate size 12 mm and safe model of the first pedestal. Above table shows that value for various plate thicknesses.

### 4.6 Analysis of second pedestal



(a) (b)

**Fig.8**. model and the boundary condition of actual model

Second pedestal is attached to the upper side of first pedestal bolted on a plate fig.8 shows that combine geometry of pedestal.

Fig 8 (a) shows the combine model for first and second pedestal. There is specific arrangement of the plates with 16 bolts to hold that pedestal with robot For the analysis of combine model we use the only force reaction of second pedestal on the first pedestal at the fix point. We took only the maximum value of force reaction for further analysis. Fig 8 (b) shows the boundary condition for the combine pedestal which will add the boundary condition on the first pedestal. The length of second pedestal is the from 900 mm to 2500 mm so we analysis the different 9 cases for finding force reaction and take maximum value only to safe the model in any length that pedestal.

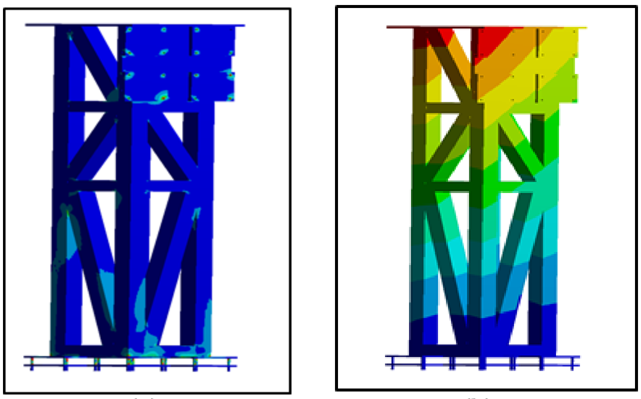
### 4.7 Analysis of actual model

Analysis of final model is done by the four different case of force and moment of the robot and maximum value of force reaction of the second pedestal gives the following results.

**Table III** analysis of final model

| Sr no | Vertical force N | Horizontal force N | Torque Nmm | Tilting moment Nmm | Equivalent stress Mpa | Total deformation mm |
|---|---|---|---|---|---|---|
| 1 | 1285 | 1032 | 7.41e5 | 5.95e5 | 198.72 | 1.057 |
| 2 | 1285 | -1032 | 7.41e5 | 5.95e5 | 141.84 | 0.3545 |
| 3 | 1285 | -1032 | 7.41e5 | -5.95e5 | 142.96 | 0.3596 |
| 4 | 1285 | 1032 | 7.41e5 | -5.95e5 | 216.11 | 1.3308 |

In fourth case gives maximum equivalent stress and maximum total deformation. the maximum stress region which is found that on the bolts so we replace the bolt instead of M10 to M12 size. After that analysis we give the safe value of the 90 Mpa stress which is below the allowable stress 150. And allowable deformation is 2mm and our deformation is 0.95 mm so we can say that the design is safe. Following fig 9 shows that equivalent stress and the total deformation.



(a) (b)

**Fig 9.**Equivalent stress and total deformation of actual model

## 5. CONCLUSIONS

Through all perform the model and analysis in ANSYS 15.0 workbench easily gives the required design solution for the robot pedestal. Combine pedestal for two robot by this method can design and which is capable for the sustain dynamic and static load of the robot By this method customized pedestal with topology optimization can be done appropriately. Modeling, meshing and FEA concluded as follows

1) With help of combine loading calculation cross sectional beam for pedestal selected.
2) Boundary conditions are applied on the first pedestal and solution is done from that solution increasing the size of base plate 12mm safe the design.
3) Attach the second pedestal to first pedestal and find the force reaction at the fix point of second pedestal which is finding at the case of clockwise tilting moment boundary condition.
4) Apply all the boundary condition final model and safe the design below 150 Mpa

## REFERENCES

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 2010.

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

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

K. 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.