DESIGN AND ANALYSIS OF ROBOT BASE

Prof.Dipika Pankaj Mahale, DR.S.N Shelke

SVIT, CHINCHOLI NASHIK

Abstract

A robot is any consequently worked machine which replaces and diminishes human exertion, however it may not look similar as a person or working in a human like way. Most robots are intended for explicit capacities inside a custom climate for performing elevated undertakings. Robots require its own exceptionally made platform structure for example base construction, uniquely developed to measure and fortitude to guarantee stability while solidly supporting the robot.in this task we investigate the platform which upholds the two robots. First and foremost, we choose the model and made on the ANSYS 15.0 workbench. After that, point fitting the model in ansys with reasonable component and apply the loading conditions like as force,moments, fix support. Then, at that point getting the consequence of absolute distortion and comparable anxieties we choose the plan is protected the model of first platform and afterward discover the power response of the second platform with that qualities safe the last model. The technique and consequence of the investigation are depicted in this paper.

Keywords—robot base; finite element analysis; optimization

I. INTRODUCTION

A robot is any thusly worked machine which replaces and reduces human effort, anyway it may not appear to be comparable personally or working in a human like manner. Most robots are planned for express limits inside a custom environment for performing raised endeavors. Robots require its own extraordinarily made stage structure for instance base development, interestingly created to gauge and mettle to ensure dependability while determinedly supporting the robot.in this undertaking we explore the stage which maintains the two robots. Above all else we pick the model and made on the ANSYS 15.0 workbench. Then fitting the model in ansys with sensible segment and apply the cutoff conditions like as forces, minutes, fix support. Then getting the result of outright twisting and practically identical nerves we pick the arrangement is ensured the model of first stage and thereafter find the force reaction of the second stage with that characteristics safe the last model. The strategy and outcome of the examination are portrayed in this paper. Load within the operation so failure cannot obtain. Fig 1 shows that the arrangement of the base for rotating robot. It shoes that tentative model of the base.by using the FEM we safe that structure with the boundary condition given by respective catalogue.

The strategies of evaluation and simulation of mechanical structures the usage of the FEA approach approves researchers and mechanical engineers to construct mathematical fashions and to analyze the static and dynamic behaviours of the structural factors without delay on the computer, and optimization calculations,simulations,etc.Approximate numerical solutions, bought thru modeling for the proposed problems, have the benefits such as it can be utilized to our bodies and actual phenomena, regardless of their complexity are converge to options of proposed issues (results can also be bought with the preferred accuracy) additionally you can view the pictures, charts, graphs - intuitive and greater numerous than in the case of genuine solutions. approves to achieve an answer in a realistic time with the economically advantage.

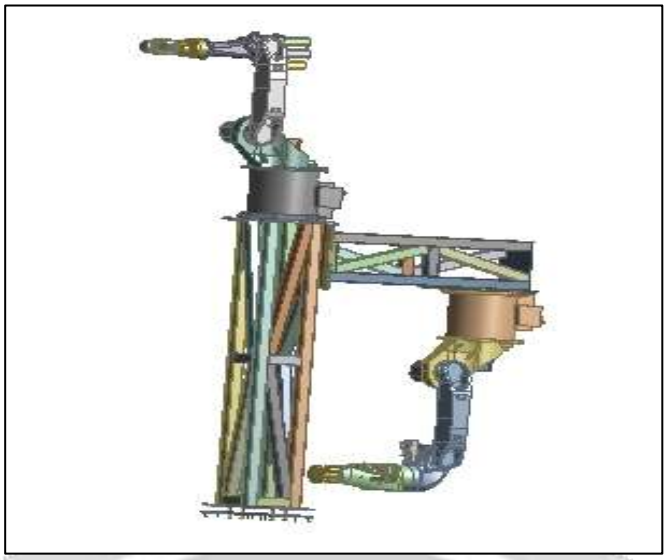


Fig 1 Robot Base

II. METHODOLOGY

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

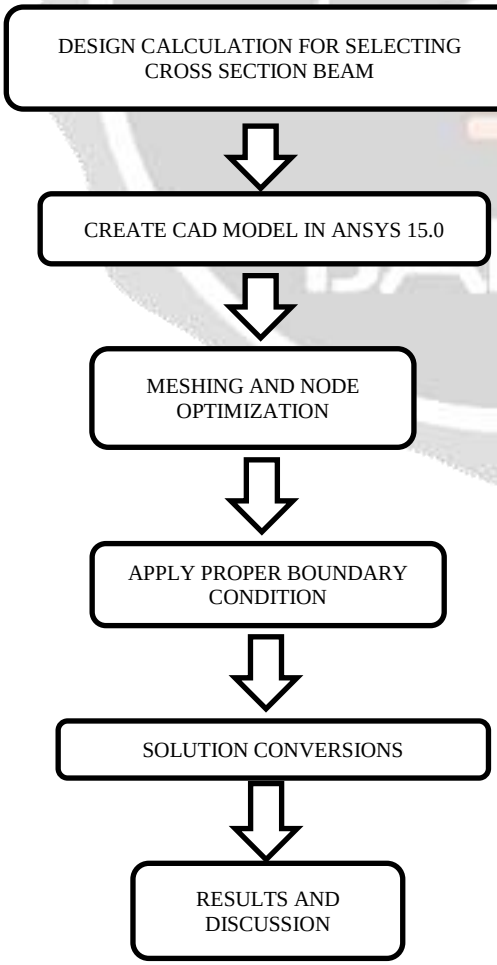


Fig2. Flowchart of methodology

III. FINITE ELEMENT ANALYSIS

The Finite Element analysis rotating robot base is to be done with help of ANSYS Workbench 16.0. The purpose of analysis was to evaluate the equivalent stress and total deformation observed in the in the structure within acceptance criteria.by analyzing the first base the find the reaction forces for second base and apply this forces on the first base and safe the design of structure given criteria.

A. Geometry:

Fig.3 robot base model

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

The material properties for analysis are shown in table.

Table I material properties

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

B. Meshing of mode:l

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

.

C. Boundary conditions:

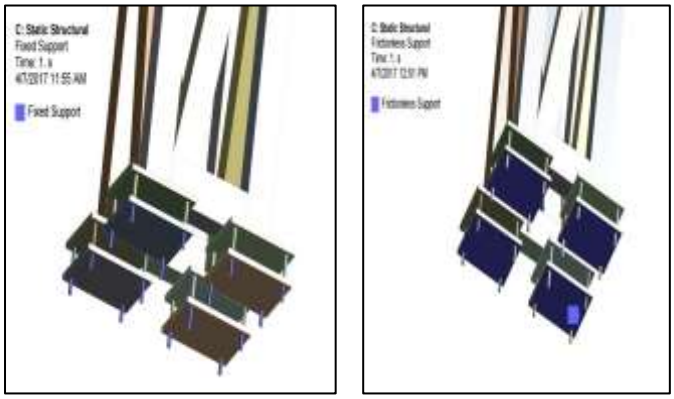
Following are the boundary condition used for analysis:

1. Forces and moment - as per the data of standard robot catalogue.

2. Fixed support – at Bolting Location

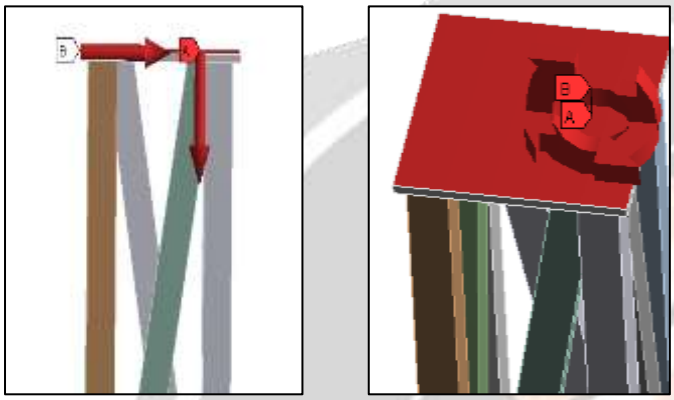
3. Frictionless support- at the base plate of base structure

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 base

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

D. Node optimization:

FEA approach offers the approximate values of solution. When we reduce the element size of mesh from course to fine nodes results are fluctuated with some difference of percentage.so we use the fine mesh for the analysis .To get suitable element size we are run the node optimization for further analysis

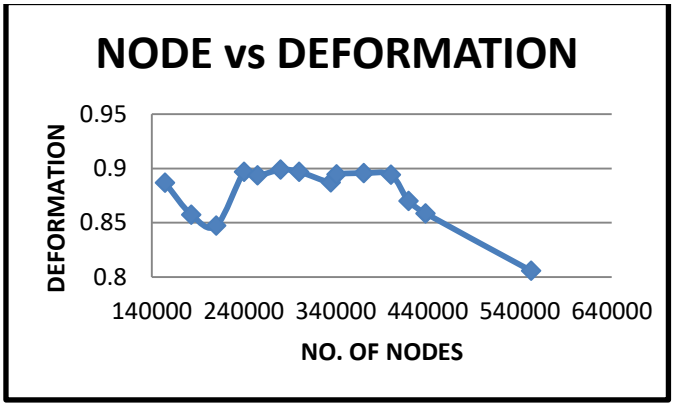


Fig.6 graph of node vs deformation

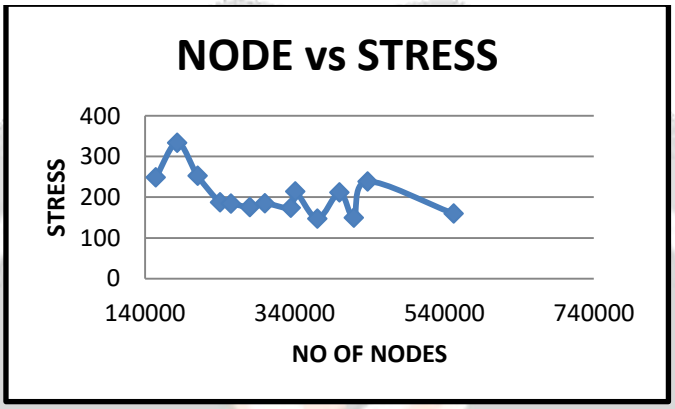


Fig.7 graph of nodes vs stress

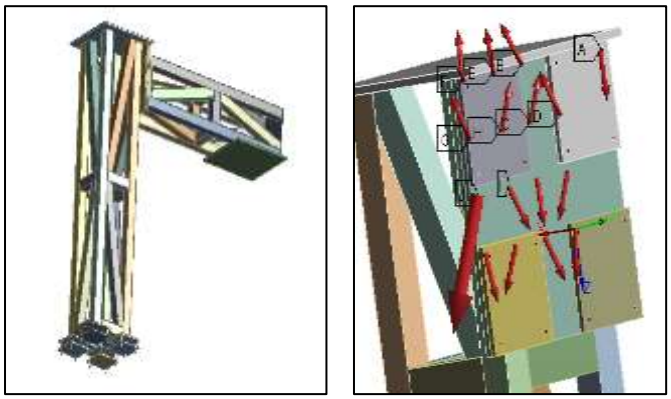
E. Plate thickness optimization:

At optimized node size around the 437000 the maximum equivalent stress is 213Mpa to avoid failure of this model it should be below the 150Mpa which is acceptance criteria. We have to find the stress concentration area with help of ansys.by observing the stress concentration found that maximum stress is produced on base plate of base.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 base. Following table shows that value for various plate thickness.

Table II result of plate thickness optimization

Sr no	Plate size mm	Equivalent stress Mpa	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

F. Analysis of second base:



(a) (b)

Fig.8. model and the boundary condition of final model

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

Fig 8 (a) shows the combine model for first and second base. There is standard location of the plates with 16 bolts to hold that base with robot For the analysis of combine model we use the only force reaction of second base on the first base at the fix point. We have taken the only the maximum value of force reaction for further analysis. Fig 8 (b) shows the boundary condition for the combine base which will add the boundary condition on the first base. The length of second base 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 base.

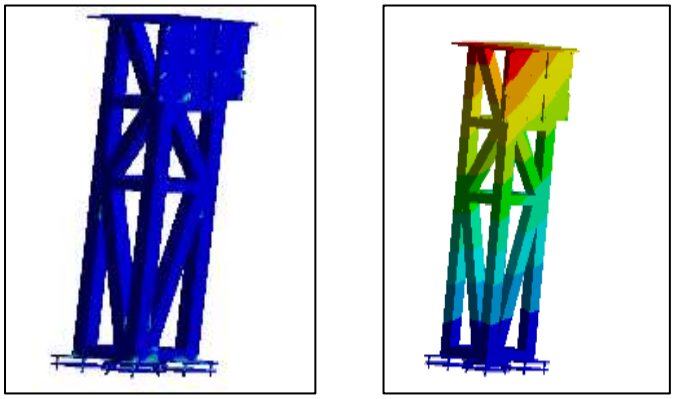
G. Analysis of final model:

Analysis of final FE model is done by the four different case of force and moment of the robot and maximum value of force reaction of the second base 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	0.35456
3	1285	-1032	7.41e5	-5.95e5	142.96	0.35969
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 will replace the bolt instead of M10 to M12 size according to that results. After that analysis we give the safe value of the 90 Mpa stress which is below the acceptance criteria 150 Mpa acceptable deformation is 2mm and our Analysis deformation is 0.95 mm so we can conclude 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 final model

IV. CONCLUSION

Through all perform the model and analysis in ANSYS 15.0 workbench easily gives the required design solution for the robot base. Combine base 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 base with topology optimization can be done appropriately.

1) With help of combine loading calculation cross sectional beam for base selected.

2) Boundary conditions are applied on the first base and solution is done from that solution increasing the size of base plate 12mm safe the design.

3) Attach the second base to first base and find the force reaction at the fix point of second base which is find 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

[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] 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.

[3] 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.

[4] 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.