IARJSET

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

2nd International Conference on Advances in Mechanical Engineering (ICAME-2016)

Amrutvahini College of Engineering, Sangamner

Vol. 3, Special Issue 1, March 2016

Analysis and Optimisation of Robot Pedestal Design using FEA

Ashwini R. Hagawane[1], Vijay L. Kadlag[2]

Postgraduate Scholar, Department of Mechanical Engineering, SVIT,Chincholi, Nashik City, India [1]

Assistant Professor, Department of Mechanical Engineering, SVIT, Chincholi, Nashik City, India[2]

Abstract: Robots requires specialized supporting structure to accurately hold the work piece during the operations. Precision made robot table and robot platforms are standard capital equipment and are required in today's high technology manufacturing companies. Most robots are designed for specific functions within a custom environment for performing elevated 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. So robotic structures are challenging because of the involving of dynamic forces. These dynamic forces further amplify themselves during emergency stop operation. Therefore robot pedestal should be well designed for operative loads and dynamic loads using estops, and also for transportation loads. Objective of the project is to design and analysis of robotic pedestal and also optimizing the structural aspects of pedestal.

Keywords: Finite Element Analysis (FEA), Vibrations, Optimisation, Case Study.

I. INTRODUCTION

Word robot was coined by a Czech novelist Karel Capek in 1920. The term robot derives from the Czech word robota, meaning forced work or compulsory service. A robot is reprogrammable, multifunctional manipulator designed to move material, parts, tools, or specialized devices through various programmed motions for the performance of a variety of tasks . A simpler version it can be define as, an automatic device that performs functions normally ascribed to humans or a machine in the form of a human. Robots require specialized supporting structure to accurately hold the workpiece during the operations. Precision made robot table and robot platforms are standard capital equipment and are required in today's high technology manufacturing companies. Most robots are designed for specific functions within a custom environment for performing elevated 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.

Fig. 1. Robot pedestal

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

IARJSET

International Advanced Research Journal in Science, Engineering and Technology
2nd International Conference on Advances in Mechanical Engineering (ICAME-2016)
Amrutvahini College of Engineering, Sangamner
Vol. 3, Special Issue 1, March 2016

Fig.1. Example of robot pedestal

A. Problem definition

Robotic structures are challenging because of the involving of dynamic forces. These dynamic forces further amplify themselves during emergency stop operation.

Therefore robot pedestal should be well designed for operative loads and dynamic loads using Estops, and also for transportation loads.

B. Objectives:

The objectives are:

- Design and analysis of robotic pedestal
- Optimizing the structural aspects of pedestal
- Optimizing the natural frequency of pedestal
- Exploring the canary design option for pedestal

II. LITERATURE REVIEW

Many researchers have explored and the progressive account of the work has been enumerated in this chapter.

A. Numerical and Experimental Study

S. Nie et al has studied a complete method for modeling and simulation for fatigue life analysis for robots with flexible joints under percussive impact forces. Though a conventional modeling method is adopted for modeling of flexible joint robots, a forced vibration solution is provided to this problem by including the impact forces generated by the percussive gun ,projecting them onto the joints pace and treating them in terms of the Fourier transform. As a result , the joint angular displacements can be solved using a standard vibration method . Then the joint stresses can be determined through Hooke's law.

JatinH.Varma has studied, a Structure can be analyzed for high loads and induced Stress values can be optimize below endurance strength of the material and deformation is reduces up to minimum level. So that Robot gun support structure can move to multiple locations quickly even causing force in tunes of 1.5 times of gravity. Jaydeep Roy and Louis L. Whitcomb has reported a comparative structural analysis of four semi-direct-drive linkages and proposed a methodology for the accurate examination and fair comparison of structural properties of disparate linkage designs for robot arms.

Randal Goldberg has reviewed a design methodology for the design of high performance arm using FEM and reported a mechanical design and supporting structural finite element structural analysis data for a new arm. Chao Yuan designed six-axis force/moment sensor. The thickness of the sensor is reduced to 12 mm, thinner than most of the multi-axis sensors used under foot and also the radius of the sensor is smaller than most of them. Cheap material and strain gages are used in this sensor to make it cheaper than other commercial sensors. The simple structure makes the fabrication of the sensor very easy. The newly modelled two part structure makes the sensor have independent adjustable sensitivities of different force components for different applications, in this paper, we just presented a special

IARJSET

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology
2nd International Conference on Advances in Mechanical Engineering (ICAME-2016)
Amrutvahini College of Engineering, Sangamner
Vol. 3, Special Issue 1, March 2016

combination of the sensitivities for Mx, My, Fz and Fx, Fy, Mz. More importantly, there is a possibility to make all the sensitivities independent and the sensor cross-coupling error free if we go further. After that, simulation results with FEM software (ANSYS) demonstrated that the design of the sensor follows the stress concentration principle. In addition, the character test results indicate that the designed sensor has good enough sensitivity, linearity error less than 0.62%F.S., hysteresis error less than 0.73%F.S., repeatability error less than 1.88%F.S. and interference error less than 3.0%F.S. Among these characteristics, some are better than some commercial sensors, some are similar with the commercial sensors, but all of them are adequate for the application of humanoid robot Sammy P feiffera, Cecilio Angulob, has developed and implemented a system for learning and executing gestures in a humanoid robot. It involves the integration of many layers of software from quite low level to very high level cogni-tive concepts. Gestures are represented via yhe use of dynamical movement primitives on the robotic platform REEM. It has been demonstrated that the use of DMPsis a very hand yway of learning motions for complex robots and it has been integrated in some experiences in aasy-to-use software. The REEM robotics currently able to learn not only gestures butal so compose tasks by learning different steps of them.

III. FINITE ELEMENT ANALYSIS (FEA)

The finite element method (FEM), sometimes referred to as finite element analysis(FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. Boundary value problems are also sometimes called field problems. The field is the domain of interest and most often represents a physical structure. The field variables are the dependent variables of interest governed by the differential equation. The boundary conditions are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field. Depending on the type of physical problem being analysed, the field variables may include physical displacement, temperature, heat flux, and fluid velocity to name only a few.[9].

IV. CASE STUDY 1

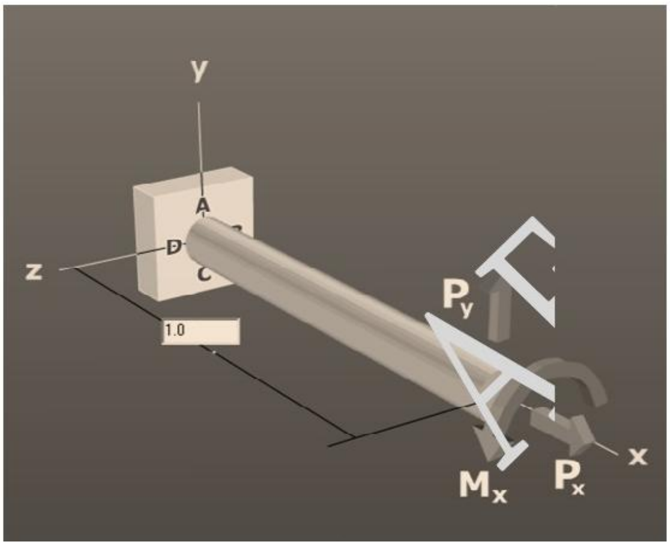

Fig. 3. Problem of case study I

CASE STUDY 1:
Combined Loading
Calculation
In Figure no 1, the inputs are as follows
Length of Shaft= 1m
Shaft Dia= 20mm
Py = 50N (bending load)
Px = 5000N (tensile load)
Mx = 140Nm (torsional moment)
Section properties for the shaft are as follows:

- OD = 20.0 mm
- ID = 0.0 mm

IARJSET

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

2nd International Conference on Advances in Mechanical Engineering (ICAME-2016)

Amrutvahini College of Engineering, Sangamner

Vol. 3, Special Issue 1, March 2016

- c = 20.0 mm / 2
  = 10.0 mm
- Area = _(20.0 mm)2 / 4
  = 314.2 mm2
- J = _(20.0 mm)4 / 32
  = 15,708.0 mm4
- I = _(20.0 mm)4 / 64
  = 7,854.0 mm4
- S = 7,854.0 mm4 / 10.0 mm
  = 785.4 mm3
- Q = (20.0 mm)3 / 12
  = 666.7 mm3

A. For stress element A (on the top of the shaft):
The force Px= 5,000.0 N creates the following stresses:
a) A uniformly distributed axial tension normal stress.
|ϭx| = |Px| / Area
= 5,000.0 N / 314.2 mm2
= 15.915 MPa

The force Py = 50.0 N creates the following stresses:
a) A linearly distributed compression normal stress due to a bending moment about the z axis. The magnitude of the normal stress is given by:
|ϭx| = |Mz y|/ I
= (1000.0 mm)(50.0 N)(10.0 mm) / 7,854.0 mm4
= 63.66 MPa

b) Although Py creates shear stress in the shaft, the transverse shear stress on element A in the y direction is zero at this location. When subjected to a shear force in the y direction, the outermost surfaces of the shaft in the y direction are free of shear stress. The concentrated torque Mx = 140.000 N-m about the x axis creates shear stress. The magnitude of the shear stress is given by:
|ς| = |T|c / J
= (140.000 N-m)(10.0 mm) / 15,708.0 mm4
= 89.127 MPa

Summary for stress element A (on the top of the shaft): The normal stresses for the combined loading can be determined by superimposing the individual cases. For stress element A (on the top of the shaft), the total normal stress acting on the element is a tension stress of 15.852 MPa. The shear stresses for the combined loading act in the positive z direction on the positive x face of the element. The magnitude of the shear stress is 89.127 MPa.
The principal stresses for the element are
ϭ1 = 137.39 MPa
and
ϭ2 = -57.81 MPa
The maximum in-plane shear stress is
ς = 195.2 MPa
and the absolute maximum shear stress equals the in-plane shear stress. This condition occurs when ϭ1 and ϭ2 have opposite signs.

B. For stress element D (on the +z side of the shaft):
The force Px= 5,000.0 N creates the following stresses:
a) A uniformly distributed axial tension normal stress.
| ϭx| = |Px| / Area
= 5,000.0 N / 314.2 mm2
= 15.915 MPa
The force Py = 50.0 N creates the following stresses:

a) Although Py creates a moment about the z axis, it produces zero flexural stress on stress element D because y = 0 at this location. In other words, element D is on the neutral axis for moments about the z axis.

IARJSET

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

2nd International Conference on Advances in Mechanical Engineering (ICAME-2016)

Amrutvahini College of Engineering, Sangamner

Vol. 3, Special Issue 1, March 2016

b) A transverse shear stress due to the 50.0 N shear force. The magnitude of the shear stress is given by:
| ς V| = |Vy|Q / I t
= (50.0 N)(666.7 mm3) / [(7,854.0 mm4) (20.0 mm)]
= 0.2122 MPa

The concentrated torque Mx = 140.000 N-m about the x axis creates shear stress. The magnitude of the shear stress is given by:
| ς T| = |T|c / J
= (140.000 N-m)(10.0 mm) / 15,708.0 mm4
= 89.127 MPa
Summary for stress element D (on the +z side of the shaft): The normal stresses for the combined loading can be determined by superimposing the individual cases. For stress element D (on the +z side of the shaft), the total normal stress acting on the element is a tension stress of 15.915 MPa. The shear stresses for the combined loading act in the negative y direction on the positive x face of the element. The magnitude of the shear stress is 88.915 MPa.

The principal stresses for the element are
Ϭ1 = 137.39 MPa
and
Ϭ2 = -57.81 MPa
The maximum in-plane shear stress is
ς = 195.2 MPa
and the absolute maximum shear stress equals the in-plane shear stress. This condition occurs when Ϭ1 and Ϭ2 have opposite signs.

Results by using ansys:

- By beam4

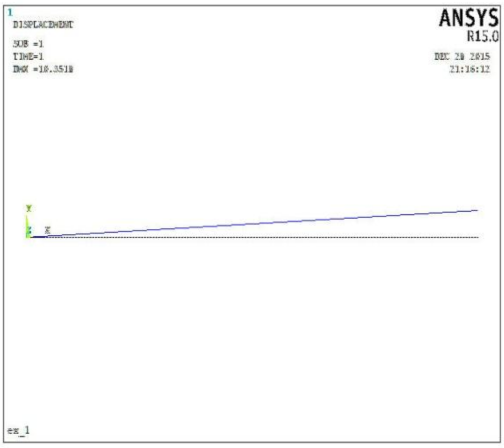

Fig .4. Deformation by beam4

- By using Beam188:

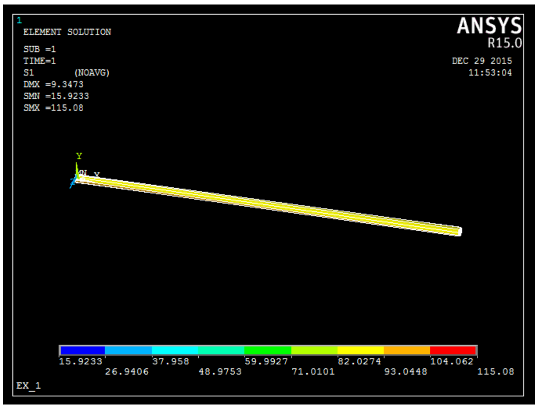

Fig. 5. Deformation by beam188

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

IARJSET

International Advanced Research Journal in Science, Engineering and Technology

2nd International Conference on Advances in Mechanical Engineering (ICAME-2016)

Amrutvahini College of Engineering, Sangamner

Vol. 3, Special Issue 1, March 2016

TABLE I
RESULT TABLE FOR CASE STUDY I

	Beam 4		Beam188	
	Theoritical	By using ansys	Theoritical	By using ansys
Ϭmax	63.332	63.332	63.332	63.332
Ϭ1	137.39	79.575	137.79	115.08

V. CASE STUDY II

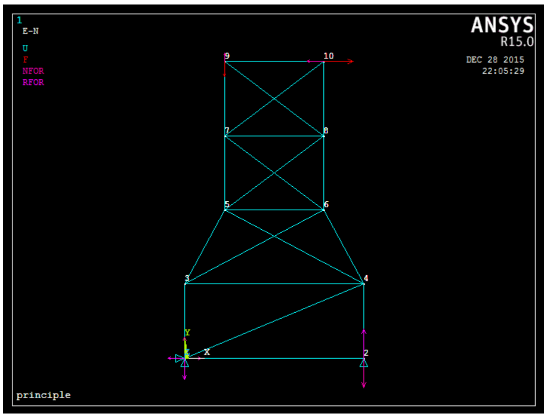

Fig. 6. Problem for case study II

To find Buckling load analytically

Buckling load= $\frac{\pi^2 EI}{L2}$, I= $\frac{bd^3}{12}$

Buckling Load at AD= $\frac{\pi^3 * 205 * 10^3 * 32552.08}{(4000)^2}$

= 4116.35N

Buckling load at CF= 4116.35N

Buckling load at EH =$\frac{\pi^3 * 205 * 10^3 * 32552.08}{(5100)^2}$

= 2532.16N

Buckling load at FH=$\frac{\pi^3 * 205 * 10^3 * 32552.08}{(2130)^2}$

= 8837.04N

Buckling load at HI=$\frac{\pi^3 * 205 * 10^3 * 32552.08}{(6280)^2}$

= 1669.98N

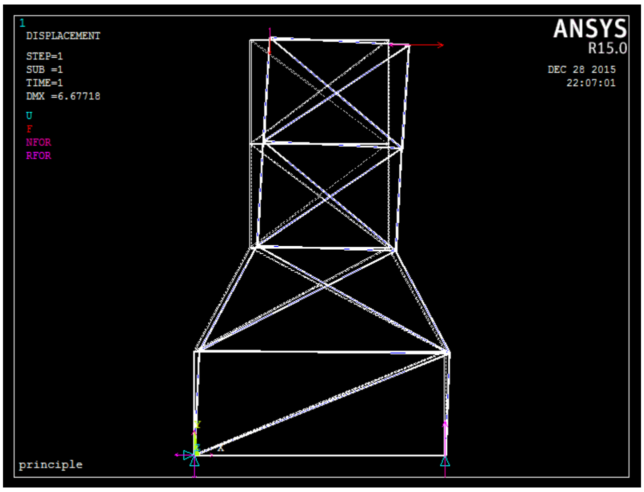

Fig. 7. Deformation of case study II in ansys

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

IARJSET

International Advanced Research Journal in Science, Engineering and Technology
2nd International Conference on Advances in Mechanical Engineering (ICAME-2016)
Amrutvahini College of Engineering, Sangamner
Vol. 3, Special Issue 1, March 2016

TABLE III: RESULT TABLE FOR CASE STUDY II

Member	Elements No	Axial load theoretical(N)	Axial load (N)MFORX analytically
AD	16	4116.35	-11525(C)
CF	12	4116.35	-13611(C)
EH	7	2532.16	377.46(T)
FH	9	8837.04	-30987(C)
HI	3	1669.98	21645(T)

Maximum buckling load by analytically found at element (4)
=-35345N comp

Result-: Actual axial load comes greater than theoretical structure is not safe under buckling.

REFERENCES

[1]. FEM Optimization for RobotStructure-Wang shijun, zhaojinjhan; IEEE ICIT'12,Bangkok, Thailand, 2002 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]. 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.
[4]. Finite Element Comparative Analysis of Two Doorframe Structures in Container Crane Zhijun Wu, kailiangLu;College of Mechanic Engineering, Tongji University, Shanghai 201804, P.R.C, National 863 plans projects. No.2009AA043000, 2011 IEEE.
[5]. Structural Dynamics Analysis of Spatial Robots With Finite Element Approach-Shiuhjer Huang, Tzyyyi Wang; Department of Mechanical Engineering, National Taiwan Institute of Technology, 43 Keelung Road, Sec. 4, Taiwan 10772, R.O.C., Computers & structures Vol. 46, No. 4, pp. 703-716, 1993.
[6]. 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 2010 IEEE, International Conference on Robotics and Automation, Albuquerque, New Mexico – April 2010, 2010 IEEE.
[7]. 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 2010.
[8]. Interval Finite Element Analysis to Structural Systems-Masahide Matsumoto, EtsuoIwaya; Proceedings of the 2010 IEEE International Workshop on Robot and Human Interactive Communication Osaka, Japan – September 27-29 2010, 2010 IEEE.
[9]. Structural Design and Analysis of a New Semi-Direct Drive Robot Arm: Theory and Experiment Jaydeep Roy, randal Goldberg; IEEE/ASME transactions on Mechatronics, Vol., 9, No. 1, March 2011, 2011 IEEE.
[10]. Finite element analysis of simple welding structure-YaodongGao, Fukuan Zhang; Inner Mongolia University of Science and Technology Institute of Mechanical Engineering, China Baotou.(2010)
[11]. A.H.Soni and M.H.F. Dado, Dynamic response analysis of 2-R robot with flexible joints. IEEE Conf. on Robotic and Automation (2010).
[12]. A. Phil and H. Asada. Rapid recursive structure redesign for improved dynamics of a single link robot.ASME Journal of Dynamics Systems, Measurement & Control (2000).
[13]. http://www.esurg.net/robotic.htm
[14]. http://www.esurg.net/robotic.htm
[15]. http://beyond-steel.blogspot.in/2012/04/harga-plat-boiler-astm-516-gr70.html http://www.ansys.com/Products/ Workflow+Technology /ANSYS+Workbench+Patform /ANSYS+Meshing /Features/Meshing+Methods:+Hexahedral