Static Analysis of Five Degrees of Freedom Articulated Robot Arm

Renish P. Vadhadiya[1] Prof. B.K. Patel[2] Chiragsinh Zala[3] Chintan C. Patel[4]
[1,3,4]PG Student [2]Assistant Professor
[1,2,3,4]Department of Mechanical Engineering
[1,2,4]L.D College of Engineering, Ahmedabad [3]R. K. University Rajkot

Abstract— Nowadays Robots play a vital role in all the activities in human life including industrial needs, the objective of this project is to analysis of a Generic articulated robot Arm. Articulated robot has been noted for application in traversing and performing manipulation in nuclear reactor facilities. Some aspects of the articulated Robot that are anticipated as useful are its small cross section and its projected ability to change elevation. In this paper concept of five-degree of freedom robot arm is discussed. This will allow a wide range of arm positions for any given target position, thus giving a great flexibility of motion. Motion can be governed by additional constraints; the paper presents an approach using modelling and a static analysis of five axis robotic arm, Robot is designed to carry 5kg payload and Analysis is performed on individual link to know exact stress pattern on each link and factor of safety value is taken as 2 and checked for safe condition.

Key word: FEM, ARA, ANSYS

I. Introduction

A. *Articulated Robot Arm:*

An articulated robot is a robot which is fitted with rotary joints. Rotary joints allow a full range of motion, as they rotate through multiple planes, and they increase the capabilities of the robot considerably. An articulated robot can have one or more rotary joints, and other types of joints may be used as well, depending on the design of the robot and its intended function [1]. The common industrial manipulator is often referred to as a robot arm, with links and joints described in similar terms. Manipulators which emulate the characteristics of a human arm are called articulated arms; the motion of articulated robot arms differs from the motion of the human arm. While robot joints have fewer degrees of freedom, they can move through greater angles. For example, the elbow of an articulated robot can bend up or down whereas a person can only bend their elbow in one direction with respect to the straight arm position. Robot is mainly defined on the parameters such as speed, load bearing capacity, accuracy, repeatability and work envelope [2].

At present, the main interest is to protect nuclear workers in highly contaminated areas or hostile environments, robots can be used in nuclear power plants to reduce human exposure not only to radiation, but also to hot, humid and oxygen-deficient atmosphere researchers in the field of robotics are proposing a great variety of robots configurations and functional capabilities to be used in nuclear power plants. Wheeled robots and tracked vehicles are the common configurations for mobile robots. The robotic system is made up of three main sub-systems: sensory head; teleportation and control panel, To carry out close inspection tasks of the vacuum vessel first wall using a long reach robot is called the "Articulated Inspection Arm" (AIA). Significant stress and high deformations in bending and torsion occur in the structure. The load depends on the articulated structure. The model has to be realistic to have a good knowledge of the end- effectors position. The model of the complete robot is the assembly of the five elementary models described before. It gives the deformation and position of the structure for any given joint position and loads. The calculation is iterative due to the non-linearities induced by the large displacements and the cumulative effect of the deformations.

B. *Finite Element Analysis:*

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. Analysis is carried out for the base arm and all the links. Base arm analysis is important, since it is the major arm that handles all the other arms and connects the robot to the base. Force and moment acting due to the weight of the arms, joints, motors and end effector. Torque due to the rotation of torsion motor located on the wrist of the arm is considered to define the external load conditions. While doing analysis mesh quality also affect the final output of analysis so great care should be taken while generating mesh.

II. Literature review

A. *Reddy B and Brioso RG. (2011) [5]*

This researches have done modelling of lower arm in the CAD software Solid Works and is evaluated structurally in the FEM software ANSYS. Parametric CAD model of robot lower arm is constructed and validate its properties with the real non-parametric model of the Industrial Robot IRB6640. It is shown that the parametric CAD model can be obtained with high accuracy. It has also been possible to validate the automated framework and the parametric design by comparing the Maximum Von Misses Stress of both arm models. Performing a Design of Experiments it was possible to obtain the functional relation between the Mesh Element Size and the FE results. The validation tests conducted further strengthen the hypothesis that tedious and repetitive design processes can be automated in order to reduce the work load of engineers.

B. Thohuraand S, ShahidulIslam MD. (2013)[6]

Finite element analysis was being carried out and shows effect of meshing quality of effects on analysis. Finite element analyses of holes are imperative because holes are used in engineering components and structures for bolts, rivets etc., and we need to know the stresses and deformation which occur near them. When generating a finite element mesh, one can expect more accurate results with more refined mesh (smaller size but larger number of elements within a confined area). However as the model gets larger (In FEA, larger model does not mean larger geometry, but rather the complexity due to the number of elements used), the computer will spend more time to generate the results of the analysis. It is often very important to minimize the computing time without a significant loss in the accuracy of the solution In this study the main target was to perform a FEM analysis on a structural problem concerning stress concentration around a hole in the middle of an infinitely long plate to validate the formula for the problem and by this to find out correct meshing of the domain.

C. Choong WH, Yeo KB. (2013) [7]

Three different materials Alloy Steel 1040, Aluminum Alloy Cast 319 and Malleable Cast Iron, have been taken and analysis has been performed. Structure with complex geometric shape has been effectively simulated and analyzed through Cosmos Works to identify the appropriate type of material and structure cross-section dimension. Table1 shows different material types for robot. Aluminum alloy is better for the low weight and high stiffness application. In this introduced approach of the dimension and material identification has improved and enhanced the lower structure design process efficiency and reduces the development time. The selected material type of alloy steel 1040 with the dimensional properties are able to reduce the robot arm repeatability error to about ±0.0705mm.

Dimension properties (mm)					Volume, V (m3)	Mass, m (kg)	Moment inertia, I (m4)	Deflection, δ (mm)	Stiffness, $k = \frac{P}{\delta}$ (N/mm)
a	B	c	d	t					
Candidates 1: Alloy Steel 1040 (E = 210GPa □=7700kg/m3)									
40	110	6	98	10	8.76E-4	6.745	2.0834E-6	0.073	6077.419
Candidates 2: Aluminum Alloy Cast 319 (E = 80GPa □=2790kg/m3)									
40	124	13	98	24	2.035-3	5.678	5.1E-6	0.0776	5667.21
Candidates 3: Malleable Cast Iron (E = 190GPa □=7300kg/m3)									
40	110	6	98	12	9.936E-4	7.253	2.24E-6	0.0744	5912,57

Table 1: Material Type for Lower Arm Structure Design [8].

III. ASSEMBLY OF ROBOT ARM

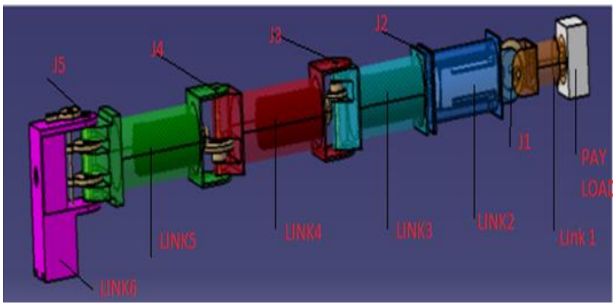

Fig. 1: Proposed design of articulated robot arm

IV. MESHING

The basic idea of FEA is to make calculations at only limited (Finite) number of points and then interpolate the results for the entire domain (surface or volume). Any continuous object has infinite degrees of freedom and it's just not possible to solve the problem in this format. Finite Element Method reduces the degrees of freedom from infinite to finite with the help of discretization or meshing (nodes and elements)

A. Decide Element Length:

Mesh size can be determined by following ways

1) Based on previous experience with a similar type of problem (successful correlation with experimental results).
2) Type of analysis: Linear static analysis could be easily carried out quickly with a large number of nodes and elements, but crash, non-linear, CFD, or dynamic analysis takes a lot of time. Keeping control on the number of nodes and elements is necessary.
3) Hardware configuration and graphics card capacity of the available computer. An experienced CAE Engineer knows the limit of the nodes that can be satisfactorily handled with the given hardware configuration.

B. Grid Impedance Test:

From above study we can conclude that meshing is an important part for carrying out FEA analysis, so this test is being carried out check best meshing size and based on that size is applied to every model, link 2 is taken for checking effect of meshing size on result, Results are compared for various mesh size.

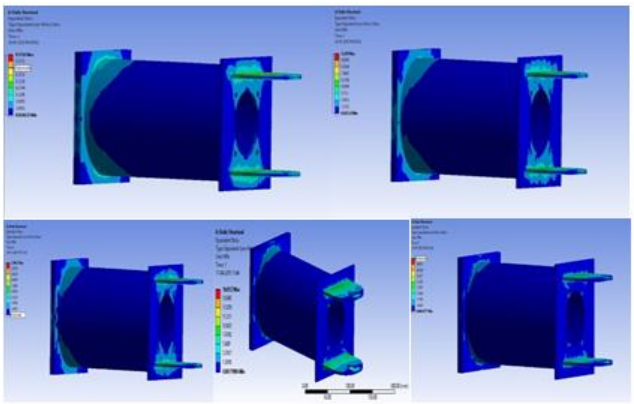

Fig. 4.1: Equivalent stress result by varying mesh size (10, 8,6,4,2 mm)

Mesh size (mm)	Equivalent stress(MPa)
10	9.3
8	11.09
6	13.3
4	16.8
2	16.9

Table 2: Comparison of stress value

From above results it is being concluded that 4mm mesh size will be good value as there is not much different with 2mm, so meshing size 4mm is taken in all parts in ansys workbench as well as in hypermesh software.

V. Static Analysis

Analysis is carried out for the base arm and all the links, base arm analysis is important, since it is the major arm that handles all the other arms and connects the robot to the base. The following are the loads considered for analysis on the arm.

1) Force due to the weight of the arms, joints, motors and end effector
2) Moment on the joint due to the weight of the arms, joints, motors and end effector
3) Torque due to the rotation of torsion motor located on the wrist of the arm

As shown in Fig 1 final assembly is prepared, it contains links, motors and gear arrangement, for analysis all weights are being considered, boundary conditions are defined from considering all weights and assuming bonded contact at joint, for exact stress result individual links analysis is carried out and that is given below. Fig 2 shows calculation of static moment at each joint.

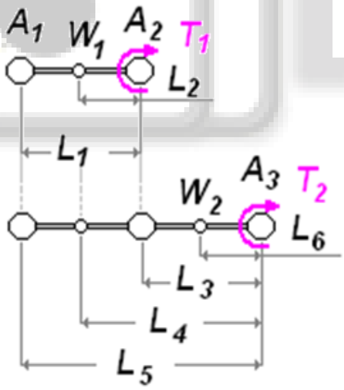

Fig. 3: static moment calculation At Joint

Static moment at joint 1 will be $M_1=A_1*L_1+W_1*L_1$, similarly it can be calculated for all joints, below table shows loading condition for all links

	Shear force N	Moment N.mm
Joint 1	75	8000
Joint 2	135	36000
Joint 3	250	77000
Joint 4	412	180000
Joint 5	550	335000

Table 3: Force and Moment Acting on each Joint

For analysis Structural steel property is assigned which is having following parameters value, analysis is carried out in ansys workbench 14.0 and also compared result with hypermesh.

Poisson's ratio = 0.3 Density = 7850 kg/m^3
Young modulus = 200 GPa Yield strength = 290 MPa

Analysis of the robot arm was analysed using dedicated software for FEM analysis. The model was exported to FEM processor i.e. in ANSYS, the geometry was updated and the structure meshed using 3Delements. Considering the load and input mentioned previously, a static analysis was carried out for the entire system, in order to obtain the main parameters of deformation, stress & fatigue. In considering static force analysis of a robot arm, all joints are first locked so that the arm becomes a structure

The static analysis comprises an assessment of the total deformation, equivalent (von Mises) stress under the loads mentioned above, max shear stress and the fatigue tool i.e. for life and damage and safety factor. An analysis of non-operational robot was done only considering the gravitational forces. The inertial forces were introduced as well, to show a complete static analysis of the operational robot.

VI. Individual Link Analysis

A. Link 1:

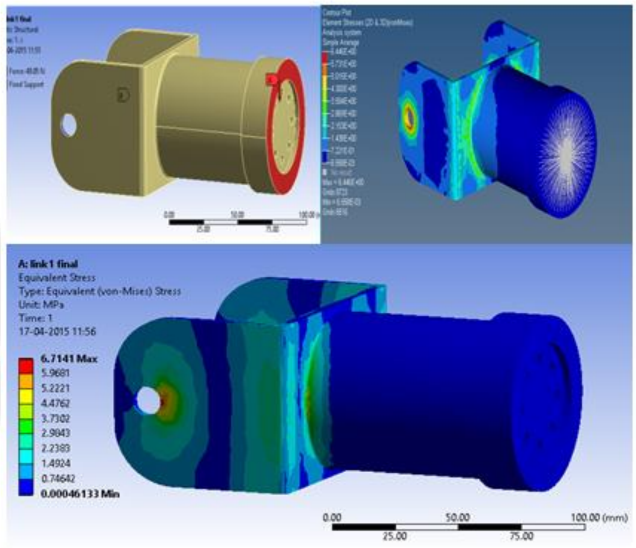

Fig. 4: Loading condition, Equivalent stress value in Hypermesh and Workbench (6.5MPa, F.O.S 44)

B. Link 2:

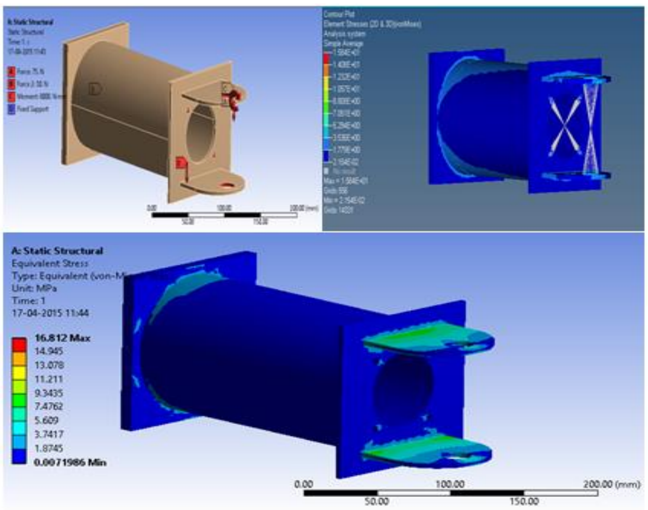

Fig. 5: Loading condition, Equivalent stress value in Hypermesh and Workbench (16 MPa, F.O.S 18)

C. Link 3:

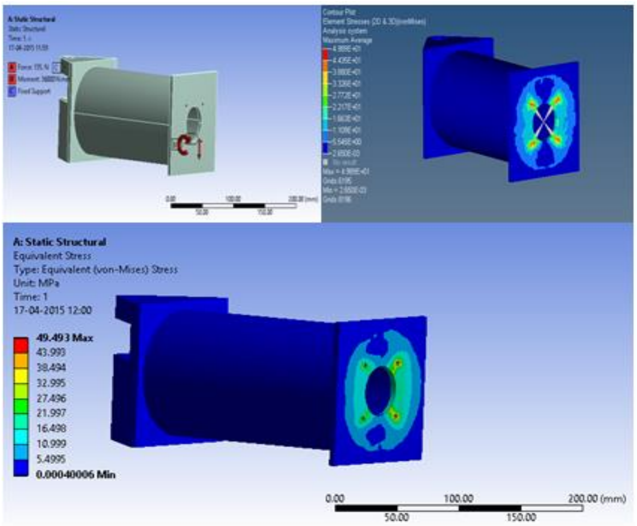

Fig. 6: Loading condition, Equivalent stress value in Hypermesh and Workbench (49 MPa, F.O.S 5)

D. Link 4:

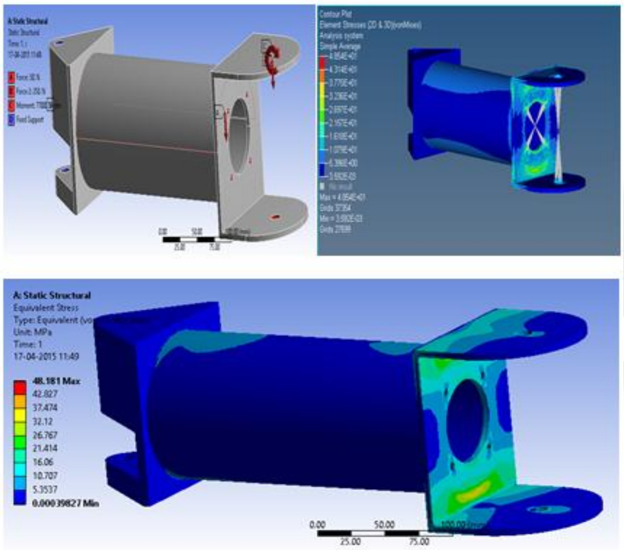

Fig. 7: Loading condition, Equivalent stress value in Hypermesh and Workbench (48MPa, F.O.S 5)

E. Link 5:

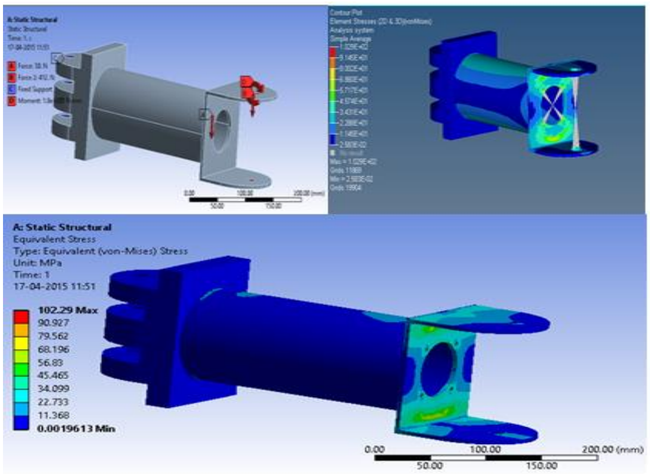

Fig. 8: Loading condition, Equivalent stress value in Hypermesh and Workbench (102 MPa, F.O.S 2.84)

F. Link 6:

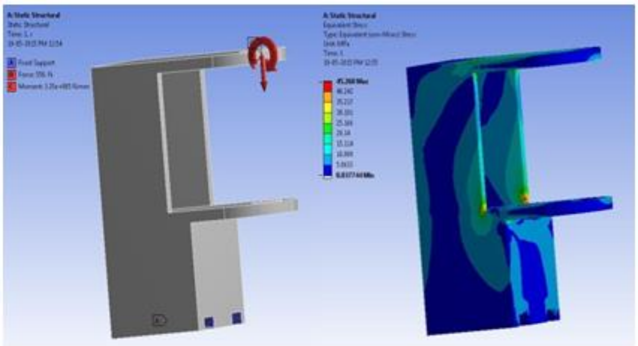

Fig. 9: Loading condition, Equivalent stress value in Hypermesh and Workbench (45 MPa, F.O.S 5)

VII. Assembly Model Analysis for Deflection

Above all results were carried out for individual links for exact stress values but it won't give total deflection of whole robot for that whole assembly is imported which consists 6 links, and another parts are suppressed for simplicity, weight of that part is putted as point mass in analysis and payload force of 49 N is applied on link 1 as shown in following image.

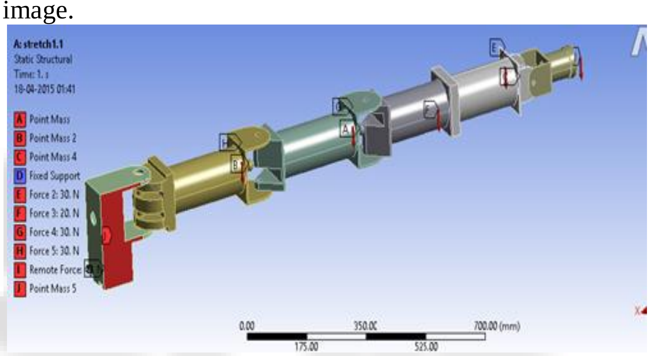

Fig. 10: boundary condition for fully stretch condition

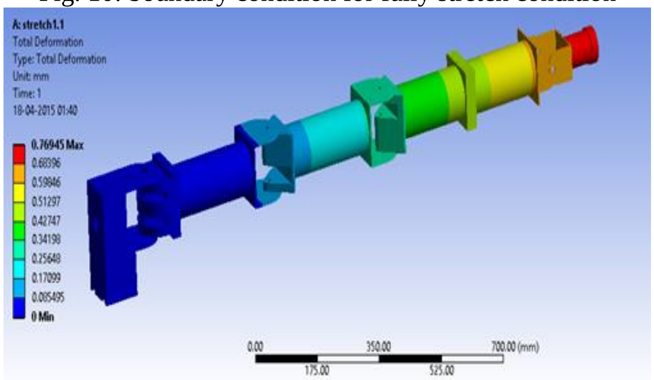

Fig. 11: Total deflection in stretch condition 0.76 mm

Allowable deflection value should be within L/750 to L/1000 here in our case L is 1.4 m so up to 1.4 mm deflection is allowed.

VIII. Conclusion and Remarks

In this paper the Static analysis of Five degrees of freedom robot arm is carried out. The static structural analysis of the Articulated Robot Arm is done and the maximum shear stress developed in the model is shown. The stresses developed and deformation occurred is also shown in the models of the Robot, Equivalent stresses and deformation of each links are shown in the fig.4 to 11 respectively. Thus, it can be seen from the results that the stress value is under limit even after considering Factor of safety so payload capacity can be increased by simple change in geometry or for same payload condition mass optimization can be followed.

REFERENCES

[1] Saha SK. Introduction to Robotics. Second edition. Delhi, India; McGraw Hill (India): 2014.
[2] Craig JJ. Introduction to Robotics-Mechanics and control. Third edition. Delhi, India; Pearson Education Inc.: 2009.
[3] Neeku SB. Introduction to Robotics- Analysis, Control and Applications. Second edition. USA; John Wiley & Sons: 2009.
[4] Mittal RK, Nagrath IG. Robotics and Control. Delhi, India; Tata McGraw-Hill: 2007.
[5] Reddy B, Brioso RG. Automated and Generic Finite Element Analysis for Industrial Robot Design. Linköping, Sweden: Linköping University Department of Management and Engineering Machine Design Division; 2011.
[6] Thohuraand S, ShahidulIslam MD. Study of the Effect of Finite Element Mesh Quality on Stress Concentration Factor of Plates with Holes. IJEIT. 2013; 3(6): 82-88p.
[7] Choong WH, Yeo KB. Structural design for a 3dof robot lower arm via computer aided engineering, Centre of Materials & Minerals. University Malaysia Sabah, Malaysia. 2013: 8-18p.
[8] Ghiorghe A. Optimization Design for the Structure of an RRR Type Industrial Robot. UPBSB. 2010; 72(4): 121-133p.
[9] Alfio N, Eula G. Optimization of a Robot for Handling Glass Plates. Department of Mechanics, Politecnico of Turin Technical University, Torino, Italy: 1-9p.
[10] Shiakolas PS, Koladiya D, Kebrle J. Optimum Robot Design Based on Task Specifications Using Evolutionary Techniques and Kinematic, Dynamic, and Structural Constraints: 1-20p.
[11] Rewatkar SS, Rahate HS, Borkar PS. Static Analysis Of Eight Degrees Of Freedom Articulated Robot Arm Under Varying Load Conditions. IJRAME. 2013; 1(7): 236-244p.