Stress and Modal Analysis of Six-Axis Articulated Robot Using ANSYS

Supriya Sahu and B. B. Choudhury

Abstract Industrial robots are used in pick and place, and various other operations in industries. So in this paper, the aim is to find out the stresses and modal analysis of different points of a six-axis industrial robot to determine its maximum shear stress, natural frequencies, and mode shapes. The optimum stress and modal analysis are done by finite element analysis (FEA) using the ANSYS workbench. For this analysis, the mesh size is taken as 0.01 mm. Different values of loads are applied on the griper to find out the maximum value of stress. For modal analysis, different cracks on the robot are considered. The modal shapes and natural frequencies for robot with crack and without crack are compared to find the weak part on the robot structure so that any design modifications can be done in order to make the robot more efficient for industrial work.

Keywords ANSYS workbench · FEA · Industrial robot · Mode shapes
Stress analysis

1 Introduction

ARISTO is a six-axis articulated robotic arm having six axes such as base, shoulder, elbow, wrist, pitch, and roll. Modal analysis is performed to find the fundamental frequencies (modes) and their associated behavior (mode shapes). This can be done by analyzing the deformation shape of structure from FEA model. High deformation area can be used for sensors placement as it can capture accurately the frequency response. Stress analysis gives the idea about the weak part of the robot

---

S. Sahu · B. B. Choudhury (✉)
Department of Mechanical Engineering, Indira Gandhi Institute of Technology, Sarang,
Dhenkanal, Odisha, India
e-mail: bbcigit@gmail.com

S. Sahu
e-mail: supriyaigit24@gmail.com

S. Fong et al. (eds.), *Information and Communication Technology for Competitive Strategies*, Lecture Notes in Networks and Systems 40,
https://doi.org/10.1007/978-981-13-0586-3_31

structure so that any design modifications if possible can be applied to get a better robotic structure.

Bhusnar and Sarawade [1] analyzed the dynamic behavior of structures like a rectangular plate, bolted lap joint and studied the natural frequencies and mode shapes using finite element analysis (FEA) and analytical methods. Chitte et al. [2] minimized the structural deformation of a six-axis robot manipulator in all the three x, y, and z directions by using calculus method on applied forces by means of which the stiffness can be improved. Ghiorghe [3] considered the criteria for minimizing the material used for building the robot structure. Finite element method has been used to determine the optimum value of design parameters. Jevan and Rao [4] analyzed the structural parameters of industrial robot, and from the analysis it has been found that the square-shaped robot arm vibrates less as compare to circular-shaped sustains. Kumar et al. [5] used FEA to analyze the natural frequencies, vibrations, and mode shapes in order to get the fracture locations in the bone by simulation. Kumar and Sambaiah [6] used finite element tool in the industrial robot for keeping the tasks in right location, with right force and torque in right time. FEA tool is also used by Nor et al. [7] to find out high stress value to determine maximum deflection in low loader structure with modeling and simulation. Pachaiyappan et al. [8] used ANSYS, the commercially available analysis tool is to analyze various critical loads acting on the articulated robot. Ristea [9] designed the control system of a motorized robot structure, along with differentiating the material such as composite and aluminum. Sridhar et al. [10] studied the behavior of industrial robot structure and used finite element method to develop the model of robot for correctness in different loading conditions.

2 The CATIA Model and Analysis

2.1 Modeling of Robot in CATIA V5

The modeling of the robot is prepared using the software CATIA V5. First of all, models of different parts of robot structure are prepared and then assembled to get the total structure. The model is then imported from CATIA into ANSYS workbench software. As the robot is of six-degree freedom structure and has to be fixed on the floor, when in use, the base surface is used as a fixed constraint.

2.2 The Finite Element Stress Analysis by ANSYS

The stress analysis is performed to test the robotic arm to withstand specific load conditions. In FEM, the model of the structure is evaluated to determine if it can handle the load conditions calculated before in the dynamic model or not. It is important to determine the stiffness of the structure. Physical analysis of the model

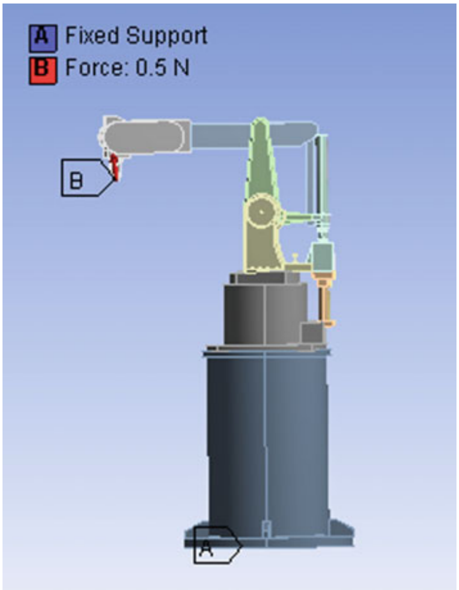


Fig. 1 Static structural setup of robot

is carried out in finite element environment. For this, the base nodes need to be constrained. For the analysis, the mesh size is taken as 0.01 mm and different loads are applied to the gripper end. The material selected is structural steel. Constraints for the assembly parts are included manually. The static structural setup of the robot is shown in Fig. 1.

For analysis of the structure, the loads applied are 0.5, 25, 50, 75, 100, and 125 N. Analysis is done to obtain shear stresses for different loads applied, and the maximum value of shear stress obtained is noted.

2.3 The Finite Element Modal Analysis by ANSYS

Vibration characteristics of the model such as natural frequency and mode shape are determined by means of modal analysis in FEA. The requirement of more capable material leads the modal analysis in systems like industrial robots. The occurrence of crack or damage in any engineering structure, rotating machines, causes premature failure and creates different operational problems. One of the criteria for fault detection is the change in dynamic behavior of structures. Due to the presence of crack, alteration of parameters such as the natural frequencies, mode shapes, and amplitude of vibration takes place.

In this present experiment, an effort has been made to explore the behavior of cracked robot. In order to achieve the three mode shapes of robot with and without crack has been analyzed.

3 Results and Discussion

3.1 Results of Stress Analysis

Optimized structural design for the structures of the industrial robots has to meet criteria regarding dimensional design and shape, material consumption and adapt this to the functional requirements. For an optimized design of the robot structure, all the aspects of industrial applications where the structure will be integrated are considered. The results achieved for maximum shear stress for each load applied are presented in Table 1.

The analysis on deformation and stress of the structure gives the idea about life, damage, and failure of robot. The shear stresses for six different gripper loads are shown in Fig. 2. The bottom part of the structure having the lowest value of shear stress is shown as dark blue color, and the top part of the structure shows the maximum value of shear stress is shown as red color. The maximum value of shear stress obtained is for 125 N which is near the gripper. The dialog box which is on the left side of the figure gives the values. The dark blue indicates the lowest value, the light blue indicates the lower value, the yellow color indicates the higher value, and red color indicates the highest value of shear stresses.

Taking these output values of shear stresses, a graph has been plotted by taking load (N) in x-axis and shear stress (Pa) in y-axis and is shown in Fig. 3.

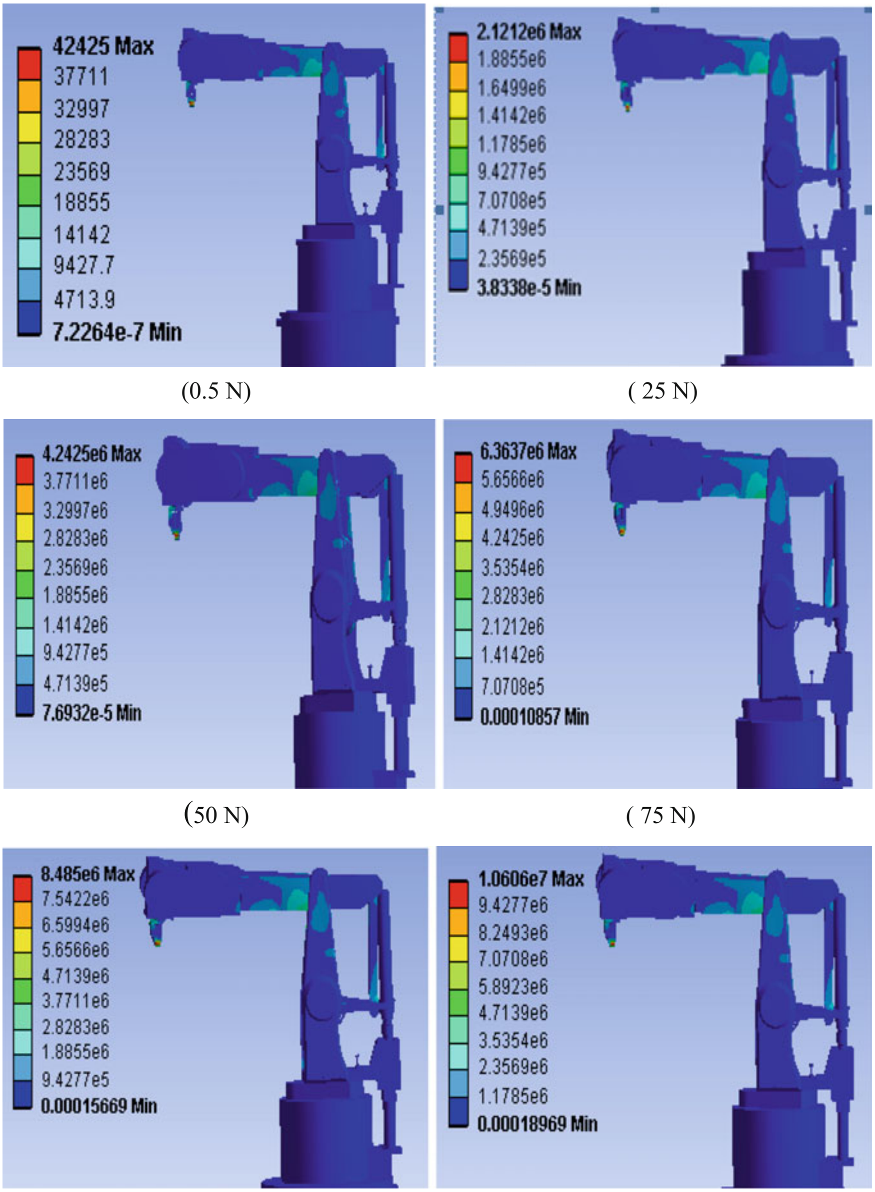
From the graph, it is seen that as load increases the shear stress increases uniformly. And for the force of 125 N, it is the highest value of $1.0606e^7$ Pa is obtained.

3.2 Results of Modal Analysis

The output frequencies of generated mode shapes at three different modes are obtained. The generated first mode shape for the robot without crack is presented in Fig. 4. The dialog box at the left side shows the values of deformations. The minimum value of deformation is shown in dark blue color which is near the base, and the maximum value of deformation is shown in red color which is near the

Table 1 Maximum shear stresses for different loads applied

SI. no.	Gripping loads (N)	Maximum shear stress (Pa)
1	0.5	42,425
2	25	$2.1212e^6$
3	50	$4.2425e^6$
4	75	$6.3637e^6$
5	100	$8.485e^6$
6	125	$1.0606e^7$



(0.5 N) (25 N)

(50 N) (75 N)

(100 N) (125 N)

Fig. 2 Shear stresses of robot model structure at various loads

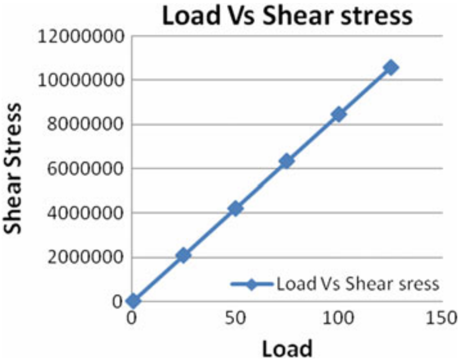
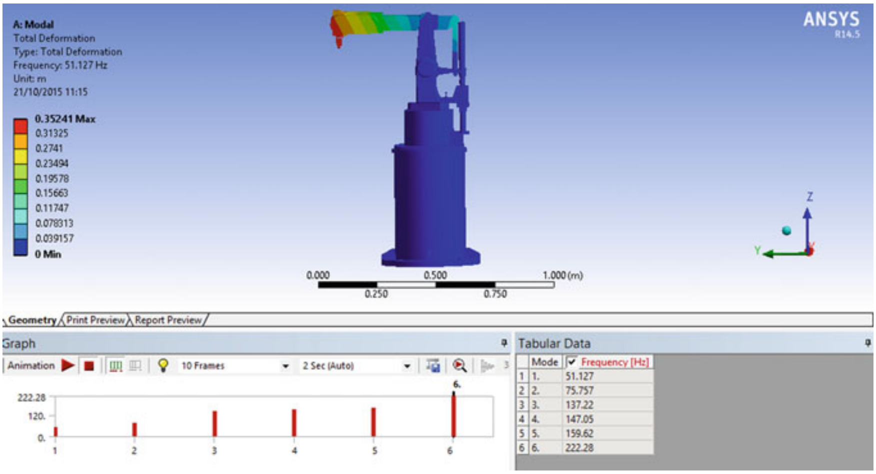


Fig. 3 Graphical presentation of load (N) versus shear stress (Pa)


A: Modal
Total Deformation
Type: Total Deformation
Frequency: 51.127 Hz
Unit: m
21/10/2015 11:15

Tabular Data

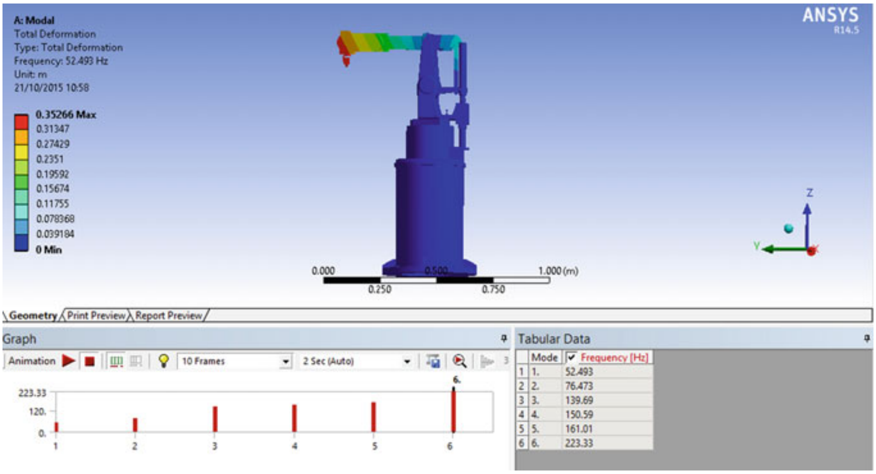
	Mode	☑ Frequency [Hz]
1	1.	51.127
2	2.	75.757
3	3.	137.22
4	4.	147.05
5	5.	159.62
6	6.	222.28

Fig. 4 Generated first mode shape for robot without crack

gripper of the color scale. The first three natural frequencies are obtained which is shown in the form of tabular data.

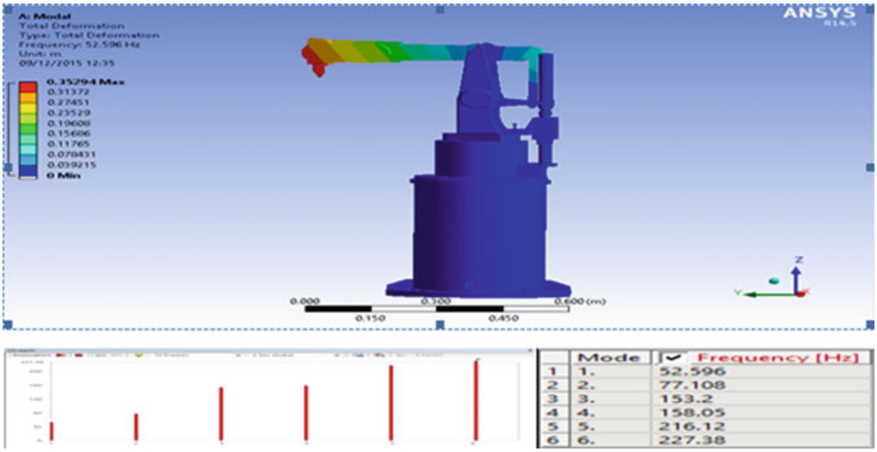
Out of three mode shapes generated, the first mode shape for robot with crack 1 is shown in Fig. 5 and for robot with crack 2 is shown in Fig. 6. From these figures, it can be observed that there are significant variations in mode shapes. The first three natural frequencies were noted down.

From these, it has been noticed that the crack had a great influence on mode shapes. In dynamic structures, the presence of crack causes vibration and affects the mechanical behavior of the structure. So the identification of cracks in the structures is a relevant issue.


A: Modal
Total Deformation
Type: Total Deformation
Frequency: 52.493 Hz
Unit: m
21/10/2015 10:58

	Mode	Frequency [Hz]
1	1.	52.493
2	2.	76.473
3	3.	139.69
4	4.	150.59
5	5.	161.01
6	6.	223.33

Fig. 5 Generated first mode shape for robot with crack 1


A: Modal
Total Deformation
Type: Total Deformation
Frequency: 52.596 Hz
Unit: m
09/12/2015 12:35

	Mode	Frequency [Hz]
1	1.	52.596
2	2.	77.108
3	3.	153.2
4	4.	158.05
5	5.	216.12
6	6.	227.38

Fig. 6 Generated first mode shape for robot with crack 2

Table 2 Frequencies for different mode shapes for robot without and with crack

Mode	Frequency without crack	Frequency with crack 1	Frequency with crack 2
1	51.123	52.528	52.596
2	137.22	139.7	153.2
3	159.62	161.06	216.12

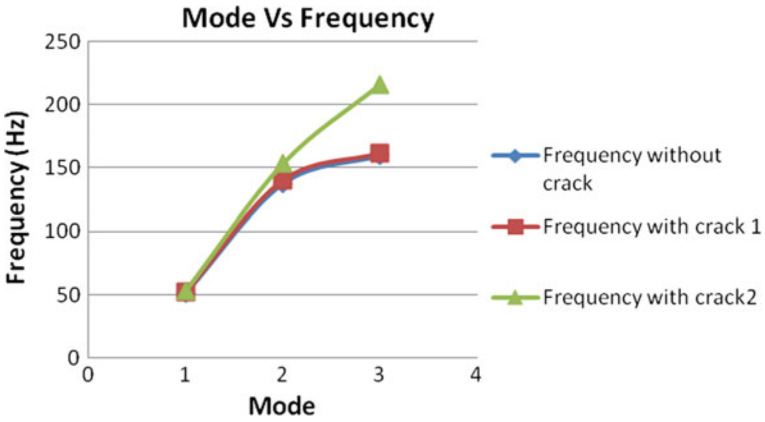


Fig. 7 Graphical presentation of mode versus frequency (Hz)

Output frequencies of ANSYS workbench for the robot with crack and without crack have been given in Table 2, and it is found that due to the presence of crack the frequencies for the three different modes increases. Comparing the frequencies of robot with crack 1 and crack 2, it can be observed from the table that for crack 2, the frequencies for three mode shapes are more than crack 1.

Taking these output values of frequencies, graphs have been plotted by taking mode in x-axis and frequency in y-axis and are shown in Fig. 7. The blue line indicates the frequencies without crack, the red line which is above the blue line indicates the frequencies with crack 1, and the green line indicates the frequencies with crack 2 for three different modes of 1, 2, and 3 as presented in the graph.

From the figure, it can be noticed that with the presence of crack, frequency of vibration increases for first mode, second mode, and also for the third mode of vibration. Due to the presence of crack, it is observed that there are significant variations in mode shapes and frequencies also depend upon the location of cracks.

4 Conclusion

The stress analysis that has been performed shows that the safety factor is reduced. Therefore, in order to maintain the safety the lifting capacity would need to be reduced to 2.5 N force. It is important to analyze the robot's structural integrity after its complete fabrication and assembly to ensure the safety factor is not adversely affected.

Modal analysis using FEA was capable of giving information about all the nodal diameter or modes for the structure. Then, the frequencies so obtained can be compared with the frequencies of sources of excitation. From the analysis, it is observed that whether those frequencies lie within the range of that structure or not, which might lead to deformation or failure of the component in some later point in time.

References

1. Bhusnar, M.S., Sarawade, S.: Modal analysis of rectangular plate with lap joints to find natural frequencies and mode shapes. IOSR J. Mech. Civil Eng. (IOSRJMCE), pp. 06–14 (2016). e-ISSN: 2278-1684, p-ISSN: 2320-334X
2. Chitte, P.G., Bansode, S.S., Rathod, S.S., Motgi, N.S.: Structural and irrational analysis of six axis ARISTO robot using ANSYS. IJIRT **3**(1) (2016). ISSN: 2349-6002
3. Ghiorghe, A.: Optimization Design for the Structure of an RRR type industrial robot. U.P.B. Sci. Bull., Series D **72**(4) (2010)
4. Jeevan Rao, A.N.: Modeling and analysis of robot ARM using ANSYS. Int. J. Sci. Eng. Technol. Res. **04**(33), 6692–6697 (2015). ISSN 2319-8885
5. Kumara, A., Jaiswala H., Garga, T., Patil, P.: Free vibration modes analysis of femur bone fracture using varying boundary conditions based on FEA. Proc. Mat. Sci. 1593–1599 (2014)
6. Manoj, K.K., Sambaiah, C.H.: Modelling and study of motion control system for motorized robot arm using mat lab and analysis of the arm by using ansys. Int. J. Mech. Ind. Eng. (IJMIE) **2**(1) (2012). ISSN No. 2231–6477
7. Nor, M.A.M., Rashid, H., Mahyuddin, W.M.F.W., Azlan, M., Mahmud, J.: Stress analysis of a low loader chassis. Proc. Eng. 995–1001 (2012)
8. Pachaiyappan, S., Balraj, M.M., Sridhar, T.: Design and analysis of an articulated robot arm for various industrial applications. J. Mech. Civil Eng. 42–53 (2014). e-ISSN: 2278-1684, p-ISSN: 2320-3334X
9. Ristea, A.: FEA Analysis for Frequency Behavior of Industrial Robot's Mechanical Elements, pp. 26–28. Sinaia, Romania (2011)
10. Sridhar, C., Doukasa, J., Pandremenosa, P., Stavropoulosa, P., Foteinopoulosa, G.: On an empirical investigation of the structural behavior of robots. Proc. CIRP **3**, 501–506 (2012)