TECHNICAL UNIVERSITY OF CLUJ-NAPOCA

ACTA TECHNICA NAPOCENSIS

Series: Applied Mathematics, Mechanics, and Engineering
Vol. 61, Issue III, September, 2018

JOINT STIFFNESS EVALUATION OF A SIMPLIFIED ROBOT ARM STRUCTURE BY USING FINITE ELEMENTS

Iulian LUPEA

Abstract: *The* article proposes a simple method to evaluate the stiffness of a robot joint by using finite element simulation. In this purpose a simple robot arm structure is chosen and the stiffness of an elastic joint is assessed through simulation. The first approach is by using a static analysis followed by a dynamic one. The geometry and the material model in the observed joint are well simulated while the material model of the links of the arm are distorted becoming very stiff and massless or completely constrained. The static and the dynamic approach are giving correlated results in terms of the equivalent joint stiffness.

Keywords: *joint stiffness, robot arm, finite elements, modal analysis, natural frequency.*

1. INTRODUCTION

In the article aspects related to the joint stiffness of a simplified manipulator considered with elastic joints and rigid links, are under observation.

The robot arm is considered of a simple geometry made from a bar, U or square shape, aluminum profile. Three elastic joints are obtained by removing some material of the profile like in figure 1 and at a angle of 60 degree like in figure 2. Later the bar is bent at the weakest spot where the joint is supposed to be positioned. Set the configuration of the robot arm by rotating finite elements about joint #2 and then joint #3

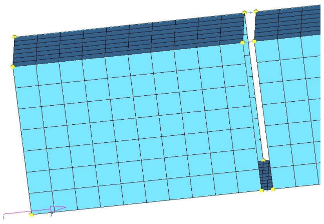

Fig. 1. Mesh, 1st joint

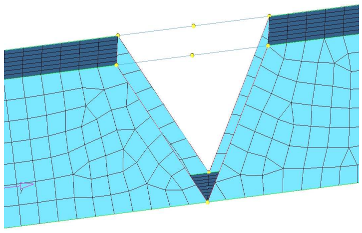

Fig. 2. Mesh of the 2nd, and the 3rd ioint

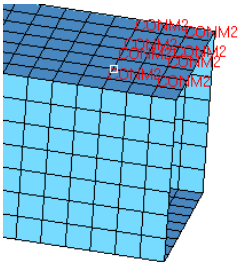

Fig. 3. Added mass or payload

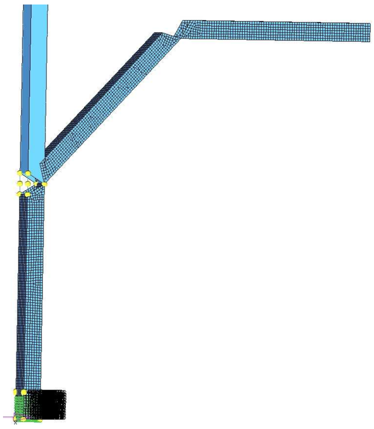

Fig. 4. Arm configuration

The arm geometry is modeled / meshed by using shell finite elements of thickness 2mm. The material model is that of the aluminum: roAl=2.8e-09[tons/mm^3], Young modulus E_Al=69e9Pa, muAl=0.33. The units proposed for the FEA model are: mm, MPa, Tones (Pa=N/m^2, MPa=N/mm^2).

The arm configuration is obtained by rotating the finite elements of the arm elements about the joints with the proper angles. In figure 4 a square shape profile arm is shown.

2. MODAL ANALYSIS OF THE ARM STRUCTURE

The modal analysis of the robot arm structure is prepared [1],[2],[3],[4],[5],[6]. A simplified structure is made from a bar square shaped transversal section. The lower end of the structure is cantilevered and the opposite end is unconstrained and with the attached mass or payload. The vibration modes in the frequency band of interest are resulting from the simulation. First three modes with movement in the arm plane are shown in figure 5. For a second simulation, U profile, the list of the first natural frequencies are shown below.

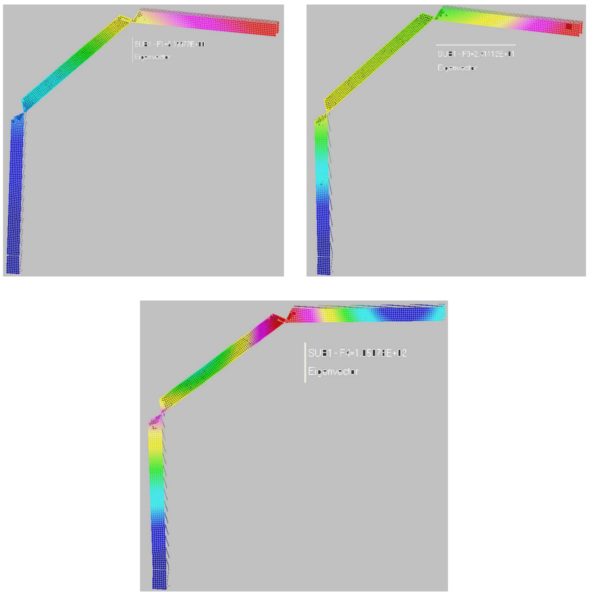

Fig. 5. Modes #1, 3, 4: 6.3Hz, 24.1Hz, 105.Hz

Subcase Mode	Frequency	Eigenvalue
1	***9.123826E+00***	3.286349E+03
2	3.305887E+01	4.314552E+04
3	***3.529078E+01***	4.916798E+04
4	***1.236178E+02***	6.032835E+05
5	1.583711E+02	9.901740E+05
6	5.093857E+02	1.024361E+07

The bold values are the frequencies of the modes with movement in the arm plane only.

3. JOINT STIFFNESS BY USING STATIC SIMULATION

The stiffness of the rotational joint#2 is found by simulation and using a static load.

The known static force F is applied in the finite element model (U profile) at the distal end of the link (l2) and perpendicular to the link (Figure 7). The observed joint #2 is deformed elastically. Two rows of shell elements both sides of the joint (Fig.6) are modeled with the proper characteristics of the aluminum. The rest of elements are with material model altered. From the displacement *disp* at the level of the force application spot, the rotational angle is determined:

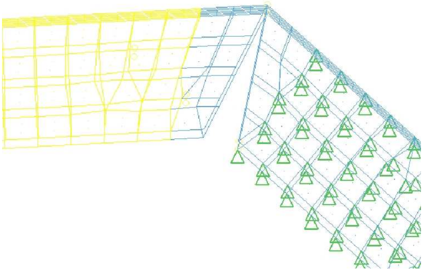

Fig. 6. Elastic joint mesh

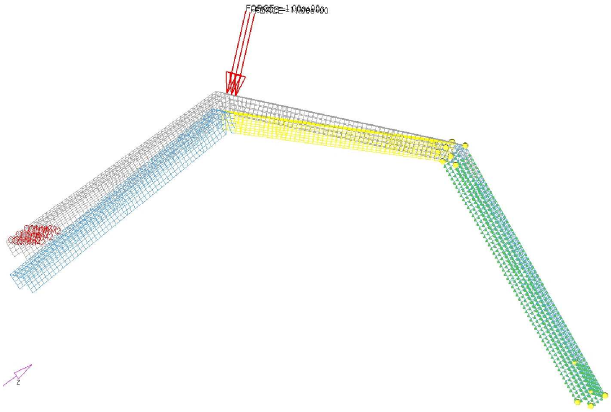

Fig. 7. Arm, static deformation of joint #2

$$\varphi_2 = disp / l2 \qquad (1)$$

where l2 is the link #2 length or the distance between the force application point and the joint #2 axis.

The associated stiffness of the observed link is derived:

$$k2 = M / \varphi_2 \qquad (2)$$

where: $M=F*l2$ and $M=k2*\varphi_2$.

The link #1 is constrained excepting the area

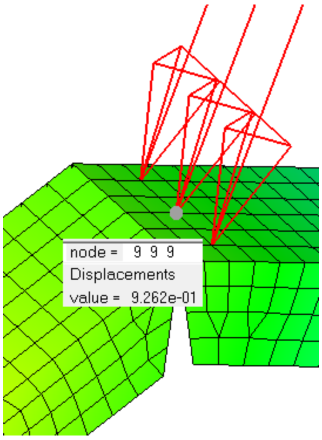

Fig. 8. Node 999 disp.

(two rows of shells) close to the elastic joint#2. As well, the link#2 (yellow) is very stiff (the material is artificially stiffened, for instance 100 times stiffer than steel), excepting two rows of shells, in order to not deform under the load F. The force F =3N is applied at the distance L=0.154 [m] from the joint, resulting from the simulation an elastic deformation *disp* of 0.926 [mm].

The applied torque M with regard to the joint axis is:

$M = F * l_2$ resulting: $M = 0.462$ [N·m]

The deformation angle φ is:

$\varphi = disp / l_2$ or: $\varphi = 0.006$ [rad]

The torsional stiffness of the second joint is: $k_2 = M/\varphi$ resulting:

$$k_{2st} = 77 \text{ [N·m]} \tag{3}$$

A total force of 1.5 N is applied instead of the previous one (3N). As a consequence of the elastic deformation the displacement in node no. 999 is 0.4631mm which is half of the displacement in the previous case, resulting the same stiffness for the joint.

The simulation procedure with the static load can be repeated in order to find the stiffness of another joint.

4. JOINT STIFFNESS BY USING DYNAMIC ANALYSIS

A second approach is applied to evaluate the joint stiffness by using a dynamic test simulation. The system will behave like a pendulum. For the same second joint stiffness evaluation, the first link is totally constrained excepting the elastic area around to the joint#2 (Figure 6) .

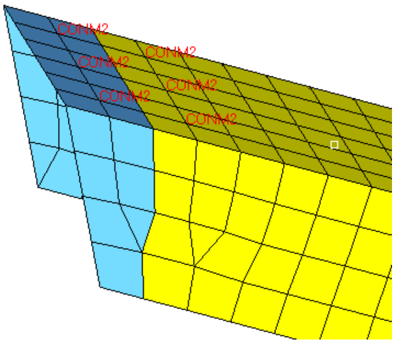

Fig. 9. Added mass - SDOF

The link #2 and the link#3 are modeled artificially to be very stiff and light (massless). A known mass ***m*** is attached to the next joint number#3 and spread in six 1D mass elements attached to six nodes of the shell finite elements. The centre of mass is in the same spot as in the static case the force is applied.

A modal analysis of the system is performed in order to find out the vibration modes. The lower vibration mode that is deforming the joint #2 about the joint axis is selected from the list. From the natural frequency f_{01} of the observed and selected mode, the stiffness of joint#2 is derived by using the equation (4) specific for the single degree of freedom system vibration:

$$\omega_0 = \sqrt{k_2 / J} \tag{4}$$

where $\omega_0 = 2\pi f_{01}$, J is the rotational inertia of the link (with the attached mass) about the second joint, oscillating about the axis of this second joint.

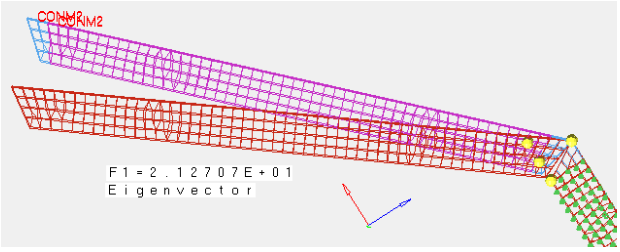

Fig. 10. First mode of vibration about joint #2

For a total mass ***m*** of 0.12kg added in the finite element simulation model, by using six mass elements of 20.000e-06 tones each, a natural frequency of f_{01}=26 Hz, is resulted. In the first mode of vibration the rigid element 3 is depicted in figure 10. The moment of inertia of the system is:

$$J = m \cdot l_2^2$$

resulting: $J = 0.002846$ [Kg·m^2]. The joint stiffness by using the dynamic simulation is:

$$k_2 = J\omega_{01}{}^2 \tag{5}$$

where ω_{01}=163.36 [rad/s].

By replacing the numerical values, the joint stiffness from the dynamic simulation results:

$$k_{2din} = 75.95 \text{ [Nm]} \tag{6}$$

For a larger total mass of 0.18e-6 tons the resulted natural frequency is smaller f_{01}=21.27 Hz and the joint stiffness results of 76.25 Nm. In Table 1, the results for the larger and a smaller attached mass are presented.

Table 1

m [tones]	f_{01}	k_{2din}

0.18 e-6	21.27	76.25
0.06 e-6	36.58	75.19

5. CONCLUSIONS

A simple method based on finite element analysis is used to evaluate the torsional stiffness of a robot joint. A simple robot arm structure with three rotation joints is considered and the stiffness of an elastic joint is assessed by simulation. The first approach is by using a static analysis; a static force is applied and the torsional stiffness is evaluated. Then a dynamic simulation is modeled and from the proper mode of vibration and the associated natural frequency the torsional stiffness of the same joint is derived. The geometry and the material model in the joint are well simulated while the material model of the links of the arm are distorted becoming very stiff and massless or completely constrained. The joint stiffness values resulted from the static and the dynamic approaches for a simple rotational joint are in good agreement. A finer mesh of the joint area can be used for comparison. The method can be extended for a more complex manipulator structure. For that case the elastic joint is modelled in detail including components like gears, harmonic drive, flexible shafts and/or a torque sensor.

6. REFERENCES

[1] He, J. and Fu, Z.F. , *Modal analysis*, Butterworth Heineman, Oxford.

[2] Lupea, I., Cormier, J., Shah, S., *FEM for impact energy absorption with Safety Plastic*, 8th International LSDYNA Users Conference, 5.57 - 5.68, 2004, Hyatt Regency, Dearborn, MI, USA.

[3] Lupea, I., Stremtan Florina-Anca, *Topological optimization of an acoustic panel under periodic load by simulation*, Acta Technica Napocensis, Series: Applied Mathematics and Mechanics, Vol. 56, Issue III, Oct. 2013, p.455-460.

[4] Lupea, I., *The Modulus of Elasticity Estimation by using FEA and a frequency response function*, Acta Technica Napocensis, Series: Applied Mathematics, Mechanics and Engineering, Vol. 57, Issue IV, Nov. 2014, pages 493-496.

[5] Lupea, I., *Updating of an exhaust system model by using test data from EMA*, Proceedings Of The Romanian Academy Series A-Mathematics Physics Technical Sciences, Information Science Volume: 14, Issue: 4, pages: 326-334, 2013.

[6] Lupea, I., Cormier, J, *Size and shape optimization of a polymeric impact energy absorber by simulation*, Materiale Plastice Volume: 44 Issue: 4 Pages: 339-344 Published: 2007.

[7] Lupea, I., Cormier, J., *Finite element model validation of a polymeric impact energy absorber*, Materiale Plastice Volume: 43 Issue: 4 Pages: 340-344 Published: 2006.

[8] ** *HyperWorks , www.altair.com*

Evaluarea rigidităţii cuplei de rotaţie a unei structuri simplificate de robot folosind elemente finite

Rezumat: In articol este prezentată o metodă simplă de evaluare a rigidităţii echivalente la torsiune a unei cuple de rotaţie aparţinând unui braţ de robot. In acest scop este folosită modelarea şi analiza cu elemente finite. Este evaluată cupla prin aplicarea unei forţe statice şi evaluarea rigidităţii la torsiune a cuplei şi printr-o metodă de simulare dinamică urmărind frecvenţa naturală aparţinând modului de vibraţie cu mişcare în planul perpendicular pe axa de rotaţie a cuplei. Rezultatele prin cele două metode sunt apropiate. Metoda se poate aplica la cuple cu geometrie mai complexă necesitând modelarea cu elemente finite în detaliu a cuplei evaluate.

Iulian LUPEA, Professor Ph.D., Technical University of Cluj-Napoca, Department of Mechanical Systems Engineering, 103-105 Muncii Blvd., 400641 Cluj-Napoca, ☎+40-264-401691, e-mail: iulian.lupea@mep.utcluj.ro ; www.viaclab.utcluj.ro