ME 613: Finite Element and Boundary Element Methods
Spring 2024, Indian Institute of Technology, Bombay
Prof. R. Ganesh

Mini Project

Due date: 07 May 2024

Instructions

As we discussed in class, we will try to replicate the Modal Analysis results for a relatively complex geometry shown in this example. This is a bonus evaluation item, worth **10** marks in your total score (out of **100**). In order for your submission to be considered for evaluation, please submit a **small report with your steps, relevant code as screenshots, as well as results and comparison with MATLAB's results**. In addition, also upload your codes that you used to solve this problem. **If any of these two items are not submitted, your submission will not be considered for evaluation. Since this is a bonus evaluation, partial marks will only be awarded sparingly.**

Problem

The objective of this problem is to perform modal analysis of the shoulder link of a Kinova Gen3 Ultra lightweight robotic arm, shown in fig. 1. You will have to write your own code to obtain the results that match with the results shown in this Example. You are given a 3D model, but as we will see through this project, the extension of 2D analysis to 3D is fairly straightforward, at least for this problem! The finite element mesh data (nodal coordinates and element connectivity) is provided as ".txt" files along with this project (RobotArm_nodes.txt and RobotArm_elecon.txt).
Note: The nodal data file (size nnodes$\times$3) provides X, Y, and Z coordinate for each node, while the elecon file (size nelmts$\times$10) provides the nodal connectivity for each element. This format is slightly different from what we used in the assignment.

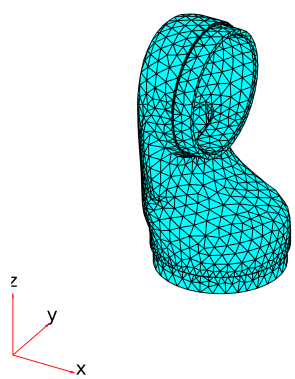

Figure 1: Finite Element Mesh of Kinova Gen3 Robotic Arm generated from MATLAB PDE Toolbox.

(a) MATLAB only has tetrahedron elements in their toolbox, and this analysis is done using second-order tetrahedron elements (10-nodes in every element). Write down the shape functions for a 10-noded tetrahedron (extend the 6-node triangle in the third dimension to get one additional vertex node $(0, 0, 1)$, and 3 additional edge nodes).

(b) Form the global stiffness and mass matrices for this given geometry using the isoparametric formulation. The relevant local mass and stiffness matrices in physical coordinates are:

$$\mathbf{K}^e = \int_{\Omega_e} \mathbf{B}^T \mathbf{D} \mathbf{B} d\Omega_e,$$
$$\mathbf{M}^e = \int_{\Omega_e} \rho \mathbf{N}^T \mathbf{N} d\Omega_e.$$

$\mathbf{K}^e$ and $\mathbf{M}^e$ are of size $3n \times 3n$, where n is the number of nodes in each element (each node has 3 dof), and $\mathbf{D}$ is the constitutive relationship between stress and strain (represented as a 6×6 matrix). This matrix can be obtained by providing Young's Modulus and Poisson's ration to the code "Calculate_prop_SVK3D.m" (see eqns. (3.18) and (3.19) here: Link). $\mathbf{N}$ and $\mathbf{B}$ are represented using the shape functions $N_i, i = 1, 2, 3, \cdots, 10$ as

$$\mathbf{N} = \begin{bmatrix} N_1 & 0 & 0 & N_2 & 0 & 0 & \cdots \\ 0 & N_1 & 0 & 0 & N_2 & 0 & \cdots \\ 0 & 0 & N_1 & 0 & 0 & N_2 & \cdots \end{bmatrix}_{3\times 3n}$$

$$\mathbf{B} = \begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & 0 & \frac{\partial N_2}{\partial x} & 0 & 0 & \cdots \\ 0 & \frac{\partial N_1}{\partial y} & 0 & 0 & \frac{\partial N_2}{\partial y} & 0 & \cdots \\ 0 & 0 & \frac{\partial N_1}{\partial x} & 0 & 0 & \frac{\partial N_w}{\partial z} & \cdots \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & 0 & \cdots \\ 0 & \frac{\partial N_1}{\partial z} & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial z} & \frac{\partial N_2}{\partial y} & \cdots \\ \frac{\partial N_1}{\partial z} & 0 & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial z} & 0 & \frac{\partial N_2}{\partial x} & \cdots \end{bmatrix}_{6\times 3n}$$

Note: You have convert these terms to isoparametric coordinates, which follows the steps we derived in class for 2D elements. The numerical integration is done using gaussian quadrature for tetrahedrons which is provided in the file "Gausst.m" - see Link.

(d) Apply fixed essential boundary conditions on the bottom surface of the shoulder link: Search for nodes with the lowest Z-coordinate value, and set all the degrees-of-freedom (X/Y/Z) to be fixed for these nodes.

(e) Solve the eigenvalue problem and obtain the natural frequency for the first seven natural modes of vibration. If λ is the eigenvalue obtained by solving $\mathbf{Ku} = \lambda \mathbf{Mu}$, then the natural frequency is $f = \sqrt{\lambda}/(2\pi)$ Hz. Compare the results obtained from your code with that given by MATLAB, and ensure that the results match.

p.s: I hope this example gives you the confidence that you can write your own FE codes to solve relatively complex problems without too much fuss (as long as you understand the fundamentals).