

Modal Analysis Of A Stepped Bar Using Matlab & Ansys

1 Gopichand Allaka, 2 Manikanta Kotti, 3 B.Srikanth, 4 Sk Rashed, And 5 D Neehaar

1(Associate Professor, Department of Mechanical Engineering, Swarnandhra College of Engineering & Technology, Andhra Pradesh-534280)

2(IV Year, B.Tech, Department of Mechanical Engineering, Swarnandhra College of Engineering & Technology, Andhra Pradesh-534280)

3(IV Year, B.Tech, Department of Mechanical Engineering, Swarnandhra College of Engineering & Technology)

4(IV Year, B.Tech, Department of Mechanical Engineering, Swarnandhra College of Engineering & Technology)

5(IV Year, B.Tech, Department of Mechanical Engineering, Swarnandhra College of Engineering & Technology)

ABSTRACT:

Man, starting from the age of civilization, wanted to simplify the processes which he felt difficult while working out in a routine way. Now we are living in the world of technology. We have different fields of study. Engineering is one among such professions which provides students the caliber to deal with the problems in a different way. Almost all the problems of various subjects which we deal in our mechanical engineering have solutions. Finite Element Methods is a subject where we deal with the elements but not the entire body at once. But still it involves a lot of calculations like Matrix multiplications, solving higher order equations etc. We have chosen a topic in Finite Element Methods i.e. Modal Analysis of elements and are working out to get results in a simpler way.

In this paper we use MATLAB software to simplify the tiresome calculations involved in Modal Analysis of Stepped bar. The code for solving the problem chosen was written in Matlab and was tested for different inputs. Even though it is accurate and has a large number of built in functions, it is lagging applications in most of the mechanical fields. The code and the procedure we are presenting here in our paper don't exist. The program written asks the user for inputs and gives output after making necessary calculations. Finally we are also using ANSYS 11.0 workbench software for verifying the results obtained in the matlab.

KEYWORDS: ANSYS, MATLAB, Modal analysis, mode, frequency.

1. INTRODUCTION

1.1 Finite Element Method

FEM is a method for dividing up a very complicated problem into small elements that can be solved in relation to each other. The Finite Element Analysis (FEA) is a numerical method for solving problems of engineering and mathematical physics.

It is useful for problems with complicated geometries, loadings, and material properties where analytical solutions cannot be obtained. Its origin is in structural mechanics. It is routinely used in the industry for design of buildings, airframes, electric motors, automobiles, and studying various materials.

1.2 Frequency

Frequency is the number of occurrences of a repeating event per unit time. For cyclical processes, such as rotation, oscillations, or waves, frequency is defined as a number of cycles per unit time. The period, usually denoted by T , is the length of time taken by one cycle, and is the reciprocal of the frequency f :

$$T = \frac{1}{f}$$

1.3 Modal Analysis

Modal Analysis is the study of dynamic properties of structures under vibrational excitation. Modal analysis is the field of measuring and analyzing the dynamic response of structures and or fluids when excited by an input. Modal analysis is a method to describe a structure in terms of its natural characteristics which are the frequency, damping and mode shapes-its dynamic properties. In structural mechanics, it is used to determine natural modes and frequencies during free vibrations.

1.4 Mode

Modes are inherent properties of a structure. Modes are associated with structural resonance. They don't depend on forces or loads acting on the structure. Modes will change if the material properties (mass, stiffness, damping properties), or boundary conditions (mountings) of the structure change. A normal mode of an oscillating system is a pattern of motion in which all parts of the system move sinusoidally with the same frequency and with a fixed phase relation.

2. INTRODUCTION TO MATLAB

MATLAB integrates mathematical computing, visualization, and a powerful language to provide a flexible environment for technical computing. The open architecture makes it easy to use MATLAB and its companion products to explore data, create algorithms, and create custom tools that provide early insights and competitive advantages.

Mat lab (**Matrix laboratory**) is an interactive software system for numerical computations and graphics. As the name suggests, Mat lab is especially designed for matrix computations: solving systems of linear equations, computing Eigen values and eigenvectors, factoring matrices, and so forth. In addition, it has a variety of graphical capabilities, and can be extended through programs written in its own programming language.

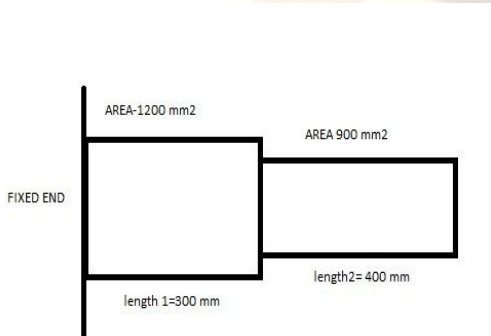
3. INTRODUCTION TO ANSYS

ANSYS, Inc. is an engineering simulation software (computer-aided engineering, or CAE) developer that is headquartered south of Pittsburgh in Canonsburg, Pennsylvania, United States. ANSYS offers a comprehensive range of engineering simulation solution sets providing access to virtually any field of engineering simulation that a design process requires. The tools in ANSYS put a virtual product through a rigorous testing procedure such as crashing a car into a brick wall before it becomes a physical object.

The **ANSYS Workbench** platform is the framework upon which the industry's broadest and deepest suite of advanced engineering simulation technology is built. With bi-directional CAD connectivity, powerful highly-automated meshing, a project-level update mechanism, pervasive parameter management and integrated optimization tools, the ANSYS Workbench platform delivers unprecedented productivity, enabling Simulation Driven Product Development.

4. PROBLEM SOLVED USING FEM:

- Q) Consider axial vibrations of the steel bar
 a) Develop the global stiffness and mass matrices.
 b) Determine the lower natural frequency and mode shape



Sol:

Given:

- Cross sectional area of first step A1=1200 mm²
- Cross sectional area of first step A2=900 mm²
- Length of first step L1= 300 mm
- Length of second step L2=400 mm
- Young's modulus E=2*10⁵ N/mm²
- Specific density ρ=7.85*10⁻⁶ kg/mm³
- We know that

$$\text{Stiffness in first step } K1 = AE/L \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} =$$

$$8*10^5 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \text{ N/mm}$$

$$\text{Stiffness in second step } K2 = AE/L \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} =$$

$$4.5*10^5 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \text{ N/mm}$$

Now add K1 & K2 to form k of 3*3 order
 Overall stiffness matrix i.e. 3*3 matrix k=

$$\begin{bmatrix} 8*10^5 & -8*10^5 & 0 \\ -8*10^5 & 12.5*10^5 & -4.5*10^5 \\ 0 & -4.5*10^5 & 4.5*10^5 \end{bmatrix}$$

N/mm

Reduced k Matrix i.e. 2*2 matrix K=

$$\begin{bmatrix} 12.5*10^5 & -4.5*10^5 \\ -4.5*10^5 & 4.5*10^5 \end{bmatrix} \text{ N/mm}$$

(As node 1 is fixed, remove 1st row & 1st column from k matrix to form K matrix)

$$\text{We know that } M1 = \rho AL/6 \begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} =$$

$$\begin{bmatrix} 0.94 & 0.47 \\ 0.47 & 0.94 \end{bmatrix} \text{ kg}$$

$$M2 = \begin{bmatrix} 0.94 & 0.47 \\ 0.47 & 0.94 \end{bmatrix} \text{ kg}$$

adding corresponding elements in M1 & M2

Overall mass matrix

$$m = \begin{bmatrix} 0.94 & 0.47 & 0 \\ 0.47 & 1.88 & 0.47 \\ 0 & 0.47 & 0.94 \end{bmatrix} \text{ kg}$$

Reduced Mass matrix i.e. 2*2 matrix

$$M = \begin{bmatrix} 1.88 & 0.47 \\ 0.47 & 0.94 \end{bmatrix} \text{ kg}$$

We know that

$$|k - xm| = 0$$

$$\begin{bmatrix} (12.5 \cdot 10^5 - x(1.88)) & (-4.5 \cdot 10^5 - x(0.47)) \\ (-4.5 \cdot 10^5 - x(0.47)) & (4.5 \cdot 10^5 - x(0.94)) \end{bmatrix}$$

=0 replace x by λ to denote universal notation

Eigen values $\lambda_1 = 1426820$

Eigen values $\lambda_2 = 166332.899$

We know that

$$\omega_1 = \sqrt{\lambda_1} = 1194.49$$

$$\omega_2 = \sqrt{\lambda_2} = 407.838$$

We know that

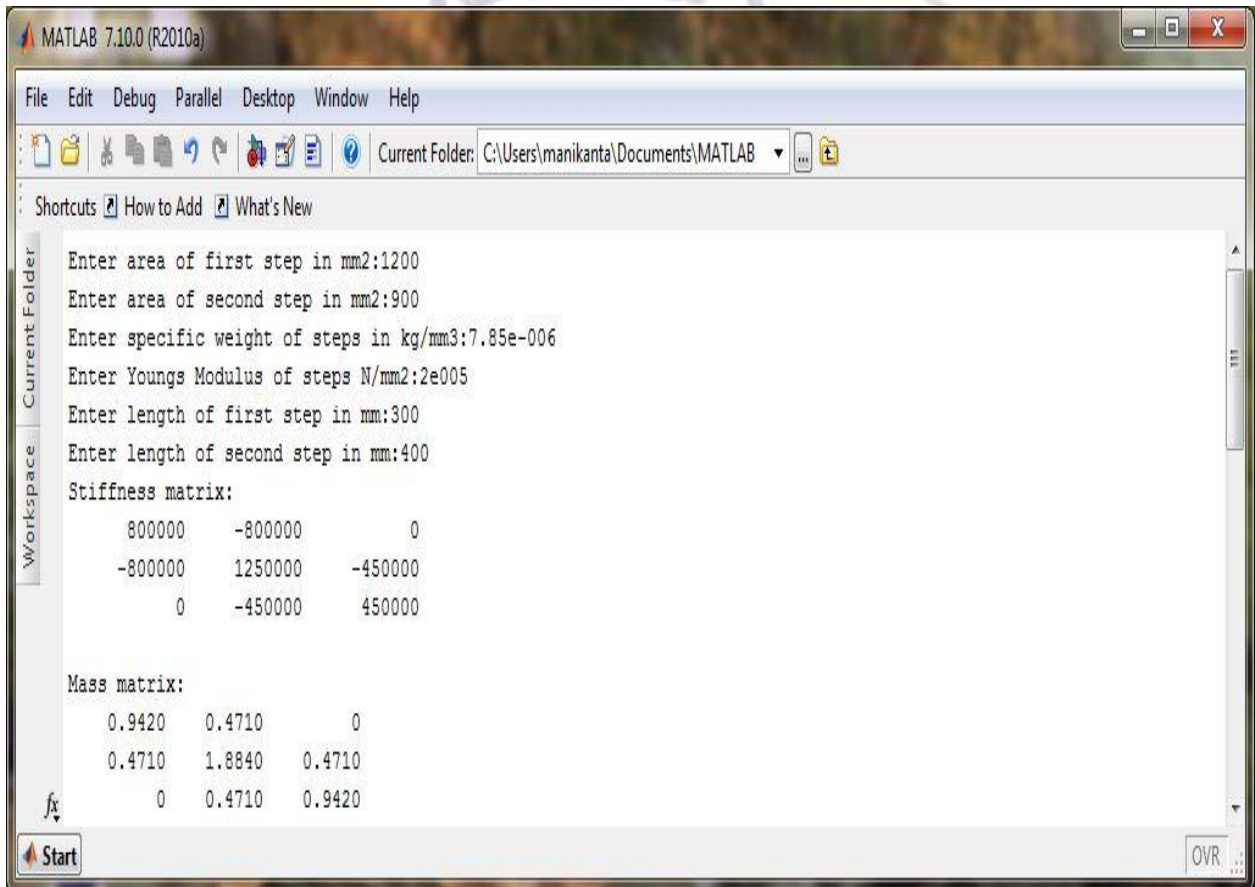
$$\omega = 2\pi f \quad f \text{ is frequency}$$

$$f_1 = 190.18 \text{ Hz} \quad f_2 = 64.9 \text{ Hz}$$

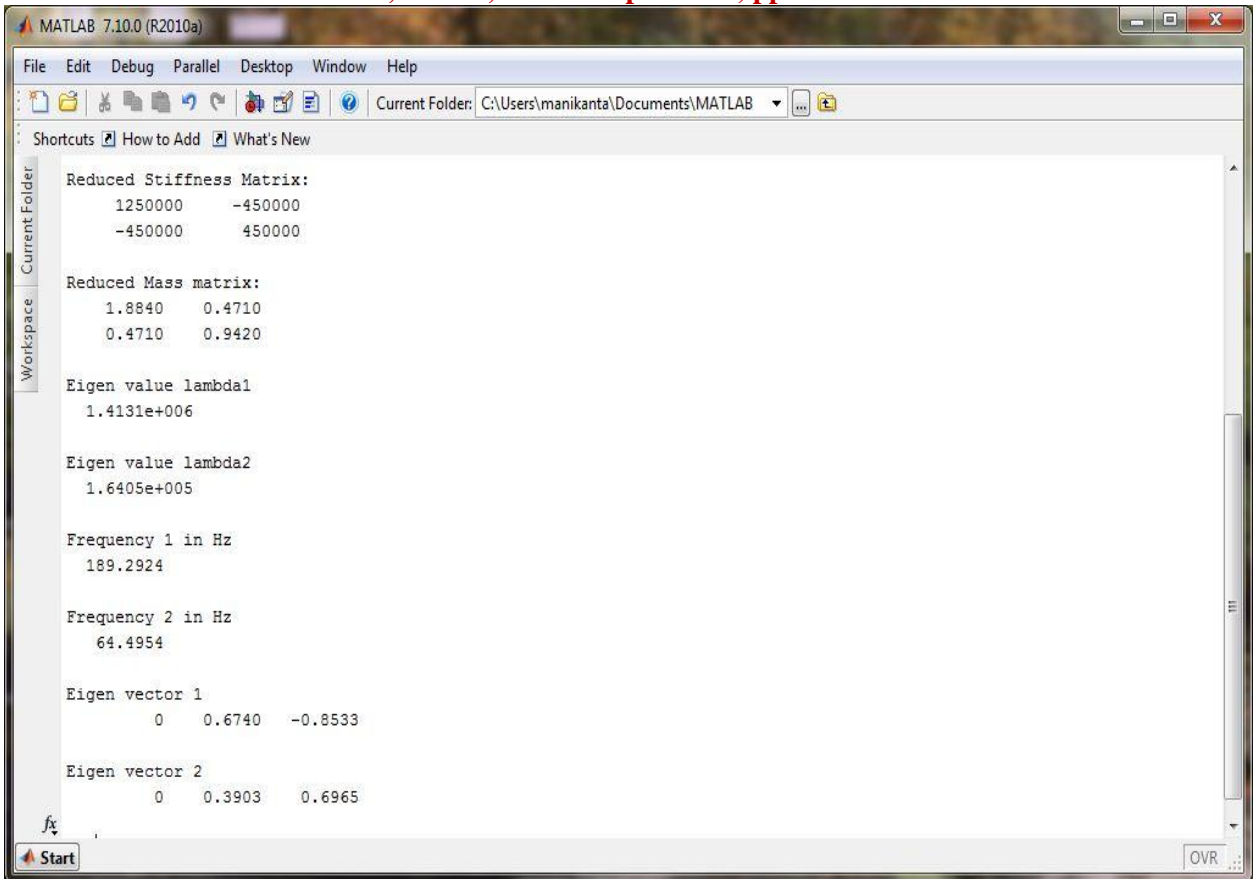
Therefore the lowest frequency is $f = 64.9 \text{ Hz}$

5. MATLAB INPUT & OUTPUT:

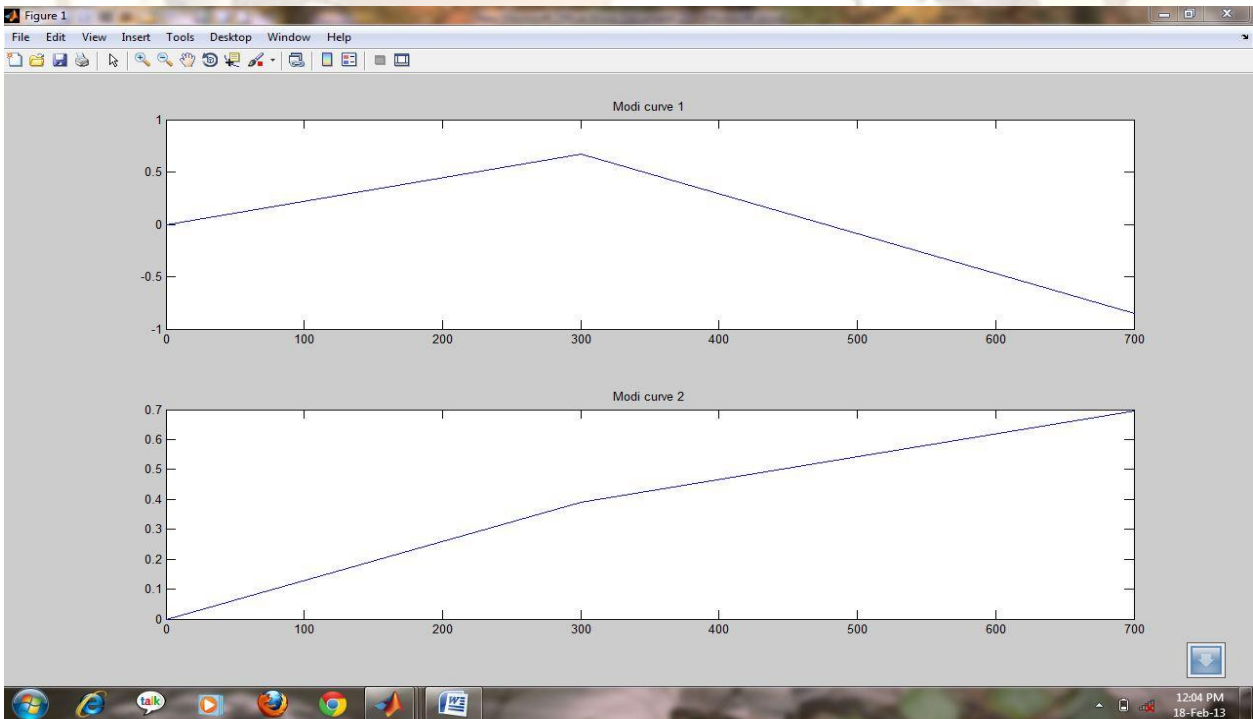
The inputs given to MATLAB are same as above solved problem and are as follows:



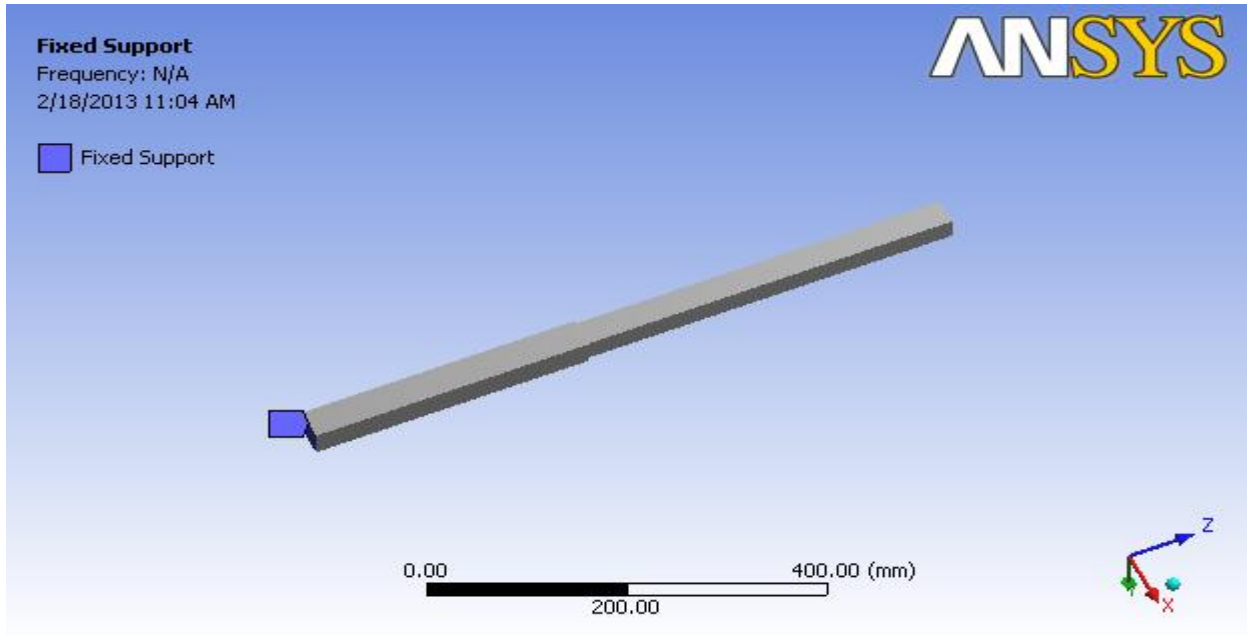
Mat lab outputs are as follows:



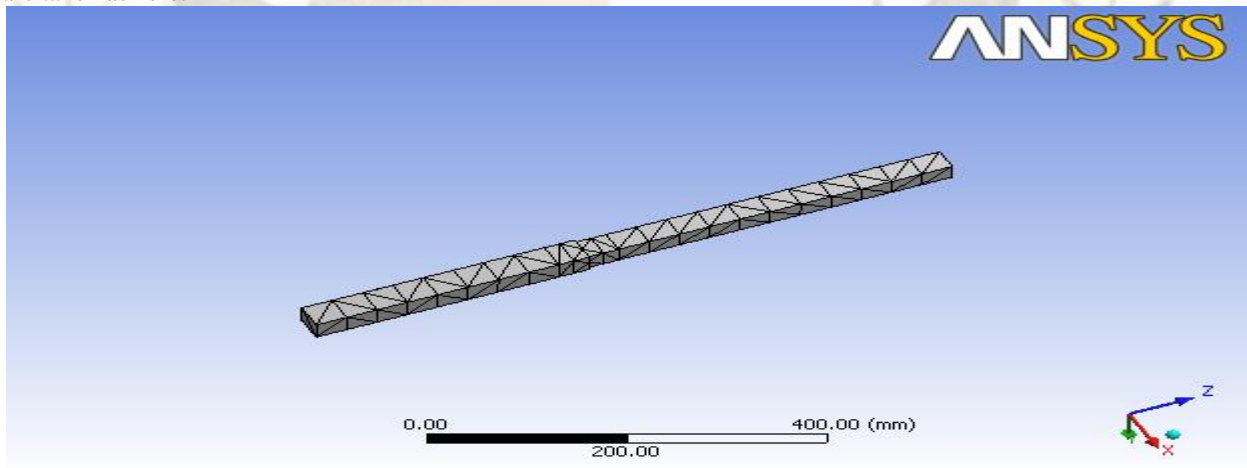
MATLAB output i.e. Mode curves are as follows:
Eigen vectors are taken on Y axis while length of steps on X axis



6. ANSYS OUTPUT:



The above fig is drawn using Pro-E software as per dimensions and was imported into Ansys 11.0 workbench. Fully developed state is chosen and structural steel is chosen as material for stepped bar. Ansys asks user for various properties regarding the material chosen. Data given for structural steel are: young's modulus $E= 2e005N/mm^2$ Poisson's ration=0.3, thermal conductivity $k= 0.1 w/mk$, specific weight $\rho= 7.85e-006 kg/mm^3$ To calculate vibrations in Z direction, dimensions in Z direction are set as free while that of X and Y directions are taken as zero.



The meshing was done as shown in the fig. After giving Initial conditions, analysis setting, loads & meshing were done to obtain final solution. The solution obtained is as follows:

Model > Modal > Solution

Mode	Frequency [Hz]
1.	64.212

This is the lowest frequency for the given problem.

7. RESULTS:

The lowest frequency obtained is as follows:

Solving using FEM	MATLAB output	ANSYS output
64.9 Hz	64.4954 Hz	64.212 Hz

7. CONCLUSION

The results obtained from MATLAB and that obtained by solving the problem using FEM were matched. Ansys 11.0 work bench results for the same problem were also close to the values obtained with an error of just around 1% which was acceptable. The MATLAB code we have written was valid for different inputs of a two stepped bar. This process of solving the problems would result in saving a lot of time, and avoiding common errors that usually occur in manual calculations. However writing MATLAB codes for solving higher order equations is difficult.

REFERENCES

- [1] Rudra Pratap, 'Getting started with MATLAB', Oxford University Press, updated for Version 7.8 (2009).
- [2] Introduction to Finite Elements in Engineering, 3rd Edition by Tirupathi R. Chandrupatla & Ashok D. Belegundu, Pearson Education, Inc.
- [3] ANSYS workbench 11.0 Tips & Tricks, by Shawn Catlin, Mallett Technology, Inc.
- [4] Introduction to MATLAB for Engineering Students, by David Houcque, North Western University, Version 1.2, (August 2005)
- [5] Modal Analysis, by Jimin He & Zhi-Fang Fu, Butterworth-Heinemann publications