

ANALYTICAL SOLUTION, FINITE ELEMENT ANALYSIS AND EXPERIMENTAL VALIDATION OF A CANTILEVER BEAM

Luis E. Monterrubio
Engineering Department
Robert Morris University

Introduction

The purpose of this work is to show how a series of labs can be used to provide students with a process that will allow them to become efficient and effective users of commercial finite element software. The series of labs starts with the study of the beam theory and analytical solutions for deflections and stresses of a cantilever beam that can be used as “exact solutions”. Then a solution for the same problem is obtained implementing the finite element method (FEM) in a Matlab code. This ensures that students understand the basic concepts of the FEM. The next step is to use different types of elements in finite element analyses (FEA) implemented in commercial software. Advantages of simple elements must be identified by students. Convergence of results towards exact values as the number of degrees of freedom in a finite element model is increased must also be noted by students. Finally, experiments are carried out to validate the FEA results. Thus, this work presents a well-integrated "lab example" that includes Matlab programming, finite element modelling and use of commercial FEM software, as well as physical verification using test equipment and sample beams.

The finite element method (FEM) is an engineering tool that allows solving several types of engineering problems. The FEM consists in discretizing a continuum into small finite elements and gives approximate solutions of boundary value problems for partial differential equations. The FEM is a way to define engineering problems in matrix form, which represent a set of linear algebraic equations.

It is relatively easy to create a mesh of finite elements using models created in computer aided engineering (CAD) software in earlier steps during the design process. More recently the geometry of the model can be obtained from a Magnetic Resonance Imaging (MRI) scanner, a Computed Tomography (CT) scanner or a laser scanner. The FEM is the major component of the Design and Analysis of Mechanical Systems course offered at Robert Morris University.

To prepare students to work efficiently and effectively in the industry using commercial FEM software it is necessary to

- Understand the theory and equations of the type of problem to be solved e.g. solid mechanics, heat transfer, vibration, etc.
- Recognize that solutions obtained by hand of simple problems can be used to check that the finite element analysis was set up correctly
- Understand the basics of the FEM, including convergence of results as the number of degrees of freedom increases
- Have knowledge and experience using different types of elements and their corresponding advantages and disadvantages
- When analysis involves a very complex geometry, it is necessary to confirm the results of the virtual prototype carrying out experimental tests with a real prototype. It is important to mention here that in the industry, the production line only starts after experimental tests confirm that the design follows all engineering specifications.

With the above goals in mind, a series of labs was developed to solve for the deflection and stress of a cantilever Euler-Bernoulli beam under a single point load as shown in Figure 1. For this problem, consider the following geometric properties: length $L=550$ mm, height $h=12.7$ mm (0.5 in) and thickness $t=63.5$ mm (2.5 in). A load equal to $m=4.53$ kg (10 lbs.) is applied 32.5 mm from its free end as shown in Figure 1. Consider the Young Modulus of the beam to be equal to $E=70$ GPa and gravity to be equal to $g=9.81$ m/s².

The approaches covered in this course to solve for the deflection and stress of the cantilever Euler-Bernoulli beam under a point load as shown in Figure 1 were

- Analytical solution. This gives a theoretical “exact” result and aims to provide all necessary background to fully understand the beam theory and its use to solve the problem. Two approaches are presented. The first approach uses the double integration of the bending moment, and the second approach uses the Rayleigh-Ritz method.
- FEM using Matlab. This approach provides a tool to assess that students have understood the basics of the FEM. How to build a global stiffness matrix, apply loads and boundary conditions, and solve for displacements and stresses.
- FEM using Abaqus student version (limited to 1000 nodes). Convergence with respect to the number of degrees of freedom is investigated using two types of elements
 - Beam elements (linear and square elements)
 - Hexahedrons (brick)

Linear beam elements B-31 have two nodes and six degrees of freedom per node. Quadratic beam elements B-32 have three nodes and six degrees of freedom per node.

- FEM Using OptiStruct. Convergence with respect to the number of degrees of freedom is investigated
 - Using Hexahedrons (brick)
 Hexahedrons Hex8 used in Hypermesh/OptiStruct are first order elements with 8 nodes per element and six degrees of freedom per node. Hypermesh/OptiStruct were used instead of Abaqus, because that was the software available at the time. The Abaqus student version is limited to 1000 nodes and is difficult to produce converged results even for this simple case.
- Experimental work
 - Measuring deflection
 - Measuring strains

The author recognizes previous works [1,2,10] that include the FEM and testing. Although there are similarities between previous works and the present work, the idea here is a little bit broader, at least with respect to the objective of helping students to become effective and efficient finite element analysts. In this work, different types of solutions are produced: analytical, FEA and experimental results. FEA include the use of different types of elements. The idea of using Abaqus and HyperMesh/OptiStruct instead of SolidWorks as in previous publications to teach FEM is very appealing to the author. This is because Abaqus and HyperMesh/OptiStruct are specialized FEA software, while SolidWorks is a mainly a CAD software with FEA capabilities.



Figure 1. Cantilever beam under concentrated load.

It must be clear also that other software and hardware than those cited in this work can be used to perform the same finite element analysis and experimental tests.

Content of the Proposed Set of Labs

The proposed set of labs described below are carried out after students have covered the basics of the FEM and are able to solve problems using spring, bar and beam elements.

Analytical Solutions

- Double Integration Method

After a review of the beam theory [9], students are shown how to derive an equation for the deflection of the beam using the double integration method [9] (as they probably did in a “strength of materials” course). Thus, considering the bending moment of the beam to be

$$M = -Fx$$

The deflection of the beam "y" can be obtained from the relation

$$M = EI \frac{d^2y}{dx^2}$$

Thus

$$EI \frac{d^2y}{dx^2} = -Fx$$

Integrating we get an equation for the slope of the beam

$$EI \frac{dy}{dx} = -\frac{1}{2}Fx^2 + C_1$$

Integrating again

$$EIy = -\frac{1}{6}Fx^3 + C_1x + C_2$$

Using the boundary conditions of the problem at $x = L$, $\frac{dy}{dx} = 0$ and at $x = L$, $y = 0$

From the first boundary condition and the equation for the slope of the beam we get

$$C_1 = \frac{1}{2}FL^2$$

From the second boundary condition and the equation for the deflection of the beam we get

$$C_2 = -\frac{1}{3}FL^3$$

The equations for the slope of the beam $\frac{dy}{dx}$ and the deflection of the beam y are

$$\frac{dy}{dx} = \frac{1}{EI} \left(-\frac{1}{2}Fx^2 + \frac{1}{2}FL^2 \right)$$

$$y = \frac{1}{EI} \left(-\frac{1}{6}Fx^3 + \frac{1}{2}FL^2x - \frac{1}{3}FL^3 \right)$$

The stress σ can be obtained from the bending moment evaluated at x and the flexural equation. Clearly in this case the maximum bending moment occurs at $x = L$

$$\sigma = \frac{My}{I}$$

Where y is the distance from the neutral axis to the point of interest and I is the second moment of area

For the cross-section of the given beam

$$I = \frac{\text{width} \cdot \text{height}^3}{12}$$

- Deflection and Stress Analysis Using the Rayleigh-Ritz Method

A second approach to calculate the deflection and stress of the beam is to use the Rayleigh-Ritz method (RRM). The RRM is an energy method that consists of carrying out a minimization (partial derivatives with respect to the unknown coefficients) of the energy terms involved in the problem. This minimization gives equations to calculate the terms of the necessary matrices to solve the problem. It is possible to obtain the deflection of a beam using the modes of vibration in the mode summation method presented by Thomson [6]. Thus, to find the displacement of the beam natural frequencies and modes of vibration must be

computed solving the generalized eigenvalue problem below:

$$[K]\{c\} - \omega^2[M]\{c\} = \{0\}$$

Where $[K]$ is the stiffness matrix, $[M]$ is the mass matrix, ω is the frequency of oscillation, $\{c\}$ is a vector of unknown coefficients and $\{0\}$ is the null vector. The stiffness matrix is obtained from the strain energy of the beam defined as [7]:

$$V = \frac{EI}{2} \int_0^L \left(\frac{\partial W(x)}{\partial x^2} \right)^2 dx$$

Where EI is the flexural rigidity of the beam. Similarly, the mass matrix of the beam is obtained from the minimization of the kinetic energy of the beam defined as [7]:

$$T = \frac{\omega^2 \rho A}{2} \int_0^L W(x)^2 dx$$

Where ρ is the density of the material, A is the cross-sectional area of the beam and $W(x)$ is the deflection shape of the neutral line of the beam that can be expressed in terms of the set of admissible functions as [7]

$$W(x) = \sum_{i=1}^n a_i \varphi_i(x)$$

Where $\varphi_i(x)$ represent the set of admissible functions that in the classical Rayleigh-Ritz method satisfy the boundary conditions of the problem. For the problem at hand consider the following set of admissible functions:

$$\varphi_i(x) = x^{i+1}$$

The minimization of the strain energy gives the terms of the stiffness matrix defined as:

$$K_{i,j} = \int_0^L EI \varphi_i''(x) \varphi_j''(x) dx$$

Where the double prime indicates the second derivative with respect to x . The terms of the mass matrix are defined by

$$M_{i,j} = \int_0^L \rho A L \varphi_i(x) \varphi_j(x) dx$$

The modes of vibration can be calculated from the generalized eigenvalue problem given earlier. The eigenvalues are the square of the circular natural frequencies, while the eigenvectors give the unknown coefficients $X_{i,j}$ of the deflection shape of the neutral axis. The i th mode of vibration can be calculated using

$$\Psi_i(x) = \sum_{j=1}^n X_{i,j} \varphi_j(x)$$

Neglecting damping, the forced motion of a linear n -degree-of-freedom system is given by the non-homogeneous equation of motion [11]

$$[K]\{y\} + [M]\{\ddot{y}\} = \{F(x, t)\}$$

The response " y " can be expressed as a linear combination of the eigenvectors $\{\Psi_i\}$

$$y = \{\Psi_i\} q_i$$

Where the generalized coordinates q_i are functions of time t and can be viewed as a coordinate transformation

$$[K][\Psi]\{q\} + [M][\Psi]\{\ddot{q}\} = \{F(x, t)\}$$

If the modes are mass normalized they can be used to uncouple the equations and solve for the deflection of the beam using the equation below as defined in the work by Thomson [6]

$$[\bar{K}]\{q\} + [\bar{M}]\{\ddot{q}\} = [\bar{F}]$$

Where

$$\begin{aligned} [\bar{K}] &= [\Psi]^T [K] [\Psi] \\ [\bar{M}] &= [\Psi]^T [M] [\Psi] \\ [\bar{F}] &= [\Psi]^T \{F(x, t)\} \end{aligned}$$

because eigenvectors are orthogonal and mass normalized

The values of the diagonalized stiffness matrix $[\bar{K}]$ are the eigenvalues, while the diagonalized mass matrix $[\bar{M}]$ is a unit matrix. Thus, for a static load we get

$$q_i = F_i / \omega_i^2$$

and the overall response is obtained by transforming the responses in generalized coordinates

$$y = \{\Psi_i\}q_i$$

See Matlab code in Appendix A

FEM using Matlab.

Students have to write two different Matlab [8] codes to solve for the displacements and stresses of the beam as follows:

- a) Write a FEM code to solve the problem using two elements. The element stiffness matrices can be obtained from shape functions of the form

$$N_i(x) = a_i + b_i(x/L) + c_i(x/L)^2 + d_i(x/L)^3$$

Applying the Rayleigh-Ritz method the terms of the stiffness matrix are

$$K_{i,j} = \int_0^L E I N_i''(x) N_j''(x) dx$$

This gives the terms of matrix K given in Appendix B.

- b) Modify the Matlab code given in Appendix B. Modifications to the code are made to define the loading vector, apply boundary conditions, and solve for the displacements. Then, gather only the values corresponding to the deflection of the beam and plotting deflection against location of the nodes. This approach allows assessing the understanding of the students of the FEM, including the process needed to define the element stiffness matrices, their contribution to the global stiffness matrix, and keeping track of the location and type (translational and rotational) degrees of freedom.

FEM Using Beam Elements in Abaqus Student Version.

The procedure to carry out the finite element analysis in Abaqus [3] using beam elements consists of

- a) Defining the geometry of the beam with two lines. One line from the free end to the location where the load is applied, and a second line from the location of the load to the constrained end of the beam.
- b) Defining the profile of the beam
- c) Defining material properties (Young's modulus and Poisson's ratio).
- d) Defining a beam section
- e) Assigning the section to the beam geometry and the beam section orientation
- f) Creating an independent instance
- g) Seeding the instance with a size equal to 0.5 to create a mesh with only two elements.
- h) Create mesh using linear beam elements (B31: A 2-node linear beam in space) [Abaqus].
- i) Create a step - static general case
- j) Apply boundary conditions and loads
- k) Create and submit a job
- l) Post-processing. The data needed to write the report are: element type and description, number of elements, number of nodes, number of active degrees of freedom, location of nodes, deflection at the nodes, and stresses in the elements.
- m) Change element type to a quadratic beam element B32, create a job, submit a job, and post-process.
- n) Comments on the accuracy of both Abaqus FEA compared to exact results must also be included in the report

FEM Using Brick Elements in Abaqus Student Version.

The procedure to carry out the FEA in Abaqus using brick elements is

- a) Define the geometry of the beam and surface where load will be applied
- b) Define material properties (Young's modulus and Poisson's ratio)
- d) Define a section
- e) Assign the section to the beam
- f) Create an independent instance
- g) Seed the instance
- h) Create mesh using hexahedron elements

(C3D8R: An8-node linear brick, reduced integration, hourglass control) [Abaqus].

- i) Create a step - static general case
- j) Apply boundary conditions and loads
- k) Create and submit a job
- l) Post-processing. The data needed to write the report are: element type and description, number of elements, number of nodes, number of active degrees of freedom, location of nodes, deflection at the nodes, and stresses in the elements.

Students must address the difficulties in creating a finite element model that produces converged results using the Abaqus student version for this simple geometry. The maximum number of nodes that can be defined in Abaqus student version is 1000.

FEM Using Brick Elements in HyperMesh/OptiStruct

- a) Open HyperMesh [4] choosing OptiStruct as user Profile and import the geometry of the beam previously created in CAD and saved in an IGES file
- b) Define material properties
- d) Choose PSOLID as the property type and select the defined material
- e) Create components
- f) Create Load Collectors to apply loads and define constraint
- g) Create load case
- h) Run analysis
- i) Post-processing

Experimental Work Deflection

To measure the deflection of the beam students used a Heightmatic HDF/Mitutoyo device shown together with the cantilever beam in Figure 2. This activity is carried out in a team.

The procedure to measure the displacement with the Heightmatic HDF/Mitutoyo device is as follows:

- a) Plug in the Heightmatic HDF
- b) Turn it on

- c) Adjust to zero moving the handle at the base until a beep is produced
- d) Use the Heightmatic HDF device to ensure that the beam is completely horizontal
- e) Load the beam with masses
- f) Measure the deflection of the beam using the handle at the base until you hear a beep
- g) Write a Table with the readings of the deflection at specific points of the beam
- h) Compare measurements with exact results from the Analytical Section.

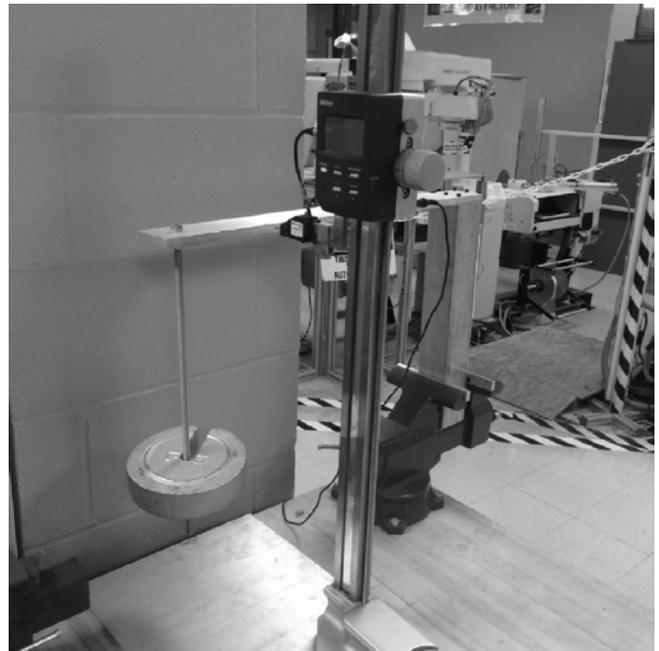


Figure 2. Set up of test to measure deflection of the beam.

Stress

Stress of the beam was measured using a $\frac{1}{4}$ bridge strain gage as shown in Figure 3, connected to a Quantum X (HBM) DAQ and the output was visualized on a laptop with catman (HBM) software installed.

Use Hooke's Law below to calculate the stress σ of the beam at the location of the strain gages and compare the results with the exact results given previously.

$$\sigma = E\varepsilon$$

where E is Young's modulus and ε is the strain.

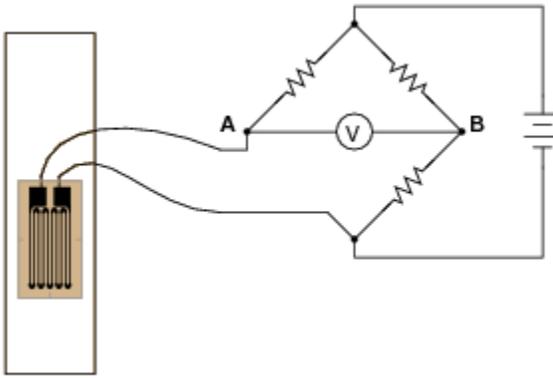


Figure 3. $\frac{1}{4}$ bridge strain gage connection.

Objectives of the Course

The main objectives of the course are

- Understand the basics of the FEM, engineering mechanics and failure prevention
- Gain experience setting up FEA in commercial finite element software, including use of different types of elements, comparing FEA results with expected results and checking convergence of the analysis refining the mesh.
- Gain experience setting up experimental tests to validate the results.

The following ABET outcomes [5] are applicable for this course according to the existing course description:

- An ability to apply knowledge of mathematics, science, and engineering
- An ability to design a system, component, or process to meet desired needs within realistic constraints such as economic, environmental, social, political, ethical, health and safety, manufacturability, and sustainability
- An ability to identify, formulate, and solve engineering problems

Assessment of the Labs

Written reports will be used to evaluate students, as well as the outcomes of their

practical work. One report is required for each lab to pace the progress of students and help providing feedback to students more frequently. The content of each report is

- Cover Page 5%
- Introduction 15%
- Procedure 20%
- Results 30%
- Conclusions 30%

To ensure learning and teaching success, it is important that students present their results in writing in an organized way together with comments and observations. Conclusions will also be used to analyze if students grasped the main concepts of the labs. In addition, project-related questions will be included in the exams to gauge student learning.

The rubric used to assess the work of the students is given in Table 1.

Conclusions

With this project, the author aims to improve students' skills to perform FEA using commercial software with labs that ensure that students understand that selecting the right element type is a very important factor in the finite element analysis. Simpler elements in many cases can lead to more accurate results and reduce computational cost. This has been shown in the labs suggested here, but also applies to modeling thin plates using shell elements in complex structures. Also by carrying out computational experiments setting up analysis using different elements, which results can be compared with exact results, students can gain confidence to become efficient and effective users of commercial FEM software. Students received an average grade of 92.4% in the hands-on projects, while the average of the final exam of the course was 76.2%. This data shows that students put more effort in the labs than in the theoretical part of the course. Experimental results were obtained with students working together to get overall better results.

Table 1. Rubric of the hands-on components of the course.

Component	Sophisticated	Competent	Not yet Competent
<p>1.2 FEM using Matlab</p> <p>ABET Outcome a</p>	<p>Codes are properly implemented showing an understanding of how to apply boundary conditions, loads and keeping track of the degrees of freedom of the system. Comments and conclusions about the differences in accuracy of the results are provided.</p>	<p>Codes solve displacements, but plots of deflection of the beam are missing or comments and conclusions do not reflect a good understanding of the results.</p>	<p>Codes do not solve the problem, comments are vague</p>
<p>1.3 FEM using beam elements in Abaqus student version</p> <p>ABET Outcome c</p>	<p>The finite element analysis must be easily completed following instructions during class. Results must be correct and more importantly comments and conclusions must state clearly how increasing the number of degrees of freedom increases the accuracy of the results.</p>	<p>There is lack of understanding about the convergence of the method as the number of degrees of freedom is increased.</p>	<p>Errors in the finite element analysis yield wrong results and conclusions.</p>
<p>1.4 FEM using brick elements in Abaqus student version</p> <p>ABET Outcome c</p>	<p>There is understanding about the convergence of the method as the number of degrees of freedom is increased. Different seeding and partition techniques are used.</p>	<p>There is understanding about the convergence of the method as the number of degrees of freedom is increased. But different seeding and partition techniques are not used.</p>	<p>There is lack of understanding about the convergence of the method as the number of degrees of freedom is increased.</p>
<p>1.5 FEM using brick elements in HyperMesh/ Optistruct</p> <p>ABET Outcome c</p>	<p>There is understanding about the convergence of the method as the number of degrees of freedom is increased.</p> <p>Results using different mesh sizes are used to ensure convergence of the method.</p>	<p>Either results using different mesh sizes did not converge or results obtained do not agree with expected results.</p>	<p>There is lack of understanding about the convergence of the method as the number of degrees of freedom is increased.</p>
<p>1.6 Experimental work – deflection and stress</p> <p>ABET Outcome e</p>	<p>There is understanding about the procedure on how to set up the tests and the results match the expected results.</p> <p>Differences between measured deflection and stresses vs exact results are explained clearly.</p>	<p>Set up the tests is not clear, reasons for differences between measured deflections and stresses vs exact results are not clearly explained.</p>	<p>Set up of the tests is wrong or missing, results don't match expected results and differences between measured deflection and stresses vs exact results are not explained.</p>

References

1. Ural, A and Yost, J., Integration of Finite Element Modeling and Experimental Evaluation in a Freshman Project, Mid-Atlantic American Society for Engineering Education Conference, 2010.
2. Howard, E., Williams, R., and Gurganus, S.C., Using Additive Manufacturing and Finite Element Analysis in a Design-Analyze-Build-Test Project, 122nd ASEE Annual Conference and Exhibition, Seattle, WA, USA., 2015.
3. ABAQUS, Student Edition, version 6.14-2, 2014, Simulia, Providence, RI.
4. HyperWorks by Altair Engineering.
5. <http://www.abet.org/eac-criteria-2014-2015/>
6. Thomson W. T. and M. D. Dahleh, Theory of vibration with applications, Upper Saddle River, N.J.: Prentice Hall, c1998.
7. Ilanko S. and Monterrubio L.E., *The Rayleigh-Ritz Method for Structural Analysis*. ISBN: 978-1-84821-638-9. Wiley-ISTE, 2014.
8. MATLAB Release 2014a, The MathWorks, Inc., Natick, Massachusetts, United States.
9. Budynas, R. G., Nisbett, J. K., & Shigley, J. E. (2011). *Shigley's Mechanical Engineering Design*. New York: McGraw-Hill.
10. Smith, N. and Davis J. L. Connecting Theory and Software: Experience with an Undergraduate Finite Element Course, 122nd ASEE Annual Conference and Exhibition, Seattle, WA, USA., 2015.
11. Boyce, DiPrima, Elementary Differential Equations and Boundary Value Problems. John Wiley & Sons, Inc., 7th ed.

Biographical Information

Luis Monterrubio joined the Robert Morris University Engineering Department as an Assistant Professor in the Fall of 2013. He earned his B.Eng. from the Universidad Nacional Autónoma de México, his M.A.Sc. from the University of Victoria, Canada, and his Ph.D. from the University of Waikato, New Zealand. All degrees are in Mechanical Engineering and both M.A.Sc. and Ph.D. studies are related with vibrations. After his Ph.D. he worked at the University of California, San Diego as postdoctoral fellow in the area of bioacoustics. He teaches dynamics, machine design, numerical methods and finite element methods. He has work for the automotive industry in drafting, manufacturing, testing (internal combustion engines—power, torque and exhaust emissions, vibration fatigue, thermo-shock, tensile tests, etc.), simulations (finite element method), and as a project manager (planning and installation of new testing facilities).

APPENDIX A

```
clear
close all
clc
format long e
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%          RAYLEIGH RITZ METHOD          %
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%          DEFLECTION SHAPE FUNCTION    %
%          The deflection shape function is: (x/L)^(i+1)          %
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%          DEFINING SOME VARIABLES      %
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Number of terms to be used in the deflection shape function
n=10;
% Number of points "nn" to be evaluated along the "X" axis.
nn=300;
% Total Length of the Beam.
TL=0.5175; % m % Location of the load
% E - Young's Modulus.
E=70e9; % Pa
% Width w and height h
w=0.0635; % m
h=0.0127; % m
% A - Area
A=w*h; % m^2
% Second Moment of Area.
I=(1/12)*w*h^3; % m^4
% rho - Density
rho=2700; % kg/m^3
% Defining the point load P.
P=-(10/2.205)*9.81; % N
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%          Defining Stiffness and Mass Matrices K and M.          %
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
for i=1:n
    for j=1:n
        K(i,j) = ((i+1)*i*(j+1)*j)/(i+j-1);
        M(i,j) = 1/(i+j+3);
    end
end
K=(K*E*I)/(TL^3);
M=M*rho*A*TL;
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%          Solving the eigenproblem          %
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% [V,D] = EIG(K,M) produces a diagonal matrix D of generalized
% eigenvalues and a full matrix V whose columns are the
% corresponding eigenvectors so that K*V = M*V*D.
[X,w] = eig(K,M);
% Changing the results for w from a square matrix to a column vector.
w=diag(w);
% Natural frequencies in Hz
Nat_Freq=sqrt(w)/(2*pi);
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%          PLOTTING THE EIGENVECTORS          %
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Describing PHI
```

```

PHI(n,nn,n)=0;
for i=1:n
    for j=1:nn
        for k=1:n
            PHI(i,j,k)=X(k,i)*((j/nn)^(k+1));
        end
    end
end
DISP=zeros(n,nn);
for i=1:n
    for j=1:nn
        for k=1:n
            DISP(i,j)=DISP(i,j)+PHI(i,j,k);
        end
    end
end
% Choosing the number of modes to be plotted.
% A maximum of eight Modes will be plotted.
% If n is smaller than eight, n Modes will be plotted.
if n>=8
    nm=8;
else
    nm=n;
end
% Building a vector containing the location of the points evaluated along
% the axis "X".
xp=0.0:(TL/nn):TL;
for i=1:nn
    f = figure('Position',[50 50 900 600]);
    plot(xp,[0,DISP(i,:)])
    str = int2str(i);
    title(['Mode ' int2str(i) ' of Vibration of a Cantilever Beam'],'FontSize',14)
    hx1=xlabel('Normalized Length of the Beam');
    hy1=ylabel('Deflection of the Beam');
end
%%%%%%%%%%
% POINT LOAD AT THE FREE END OF THE BEAM %
%%%%%%%%%%
% Applying a load at the tip end of the cantilever beam.
% The vector qgen of generalized coordinates is found to be:
for i=1:n
    qgen(i)=P*DISP(i,nn)/w(i);
end
% Transformation to the space coordinates
Usol=zeros(nn);
Usolution=zeros(nn,1);
for j=1:nn
    for i=1:n
        Usolution(j)=Usolution(j)+DISP(i,j)*qgen(i);
    end
end
Usolplot(1)=0;
for i=1:nn
    Usolplot(1+i)=Usolution(i);
end

plot(xp,Usolplot)

```

APPENDIX B

```
clear all
clc
close all
format long e
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% DEFINITION OF SOME VARIABLES
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Defining the Number of Elements of the Beam.
n=207;
% Defining the Total Length of the Beam.
TL=0.550; % [m]
% Defining the Length of each element. Elements are considered to be identical.
L=TL/n; % [m]
% E - Young's Modulus.
E=; % [Pa]
% I Second Moment of Area.
I=; % [m^4]
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Defining the stiffness matrix K of a beam element
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
K(1,1)=12*E*I/(L^3);
K(1,2)=6*E*I/(L^2);
K(1,3)=-12*E*I/(L^3);
K(1,4)=6*E*I/(L^2);
K(2,1)=6*E*I/(L^2);
K(2,2)=4*E*I/L;
K(2,3)=6*E*I/(L^2);
K(2,4)=2*E*I/L;
K(3,1)=-12*E*I/(L^3);
K(3,2)=-6*E*I/(L^2);
K(3,3)=12*E*I/(L^3);
K(3,4)=-6*E*I/(L^2);
K(4,1)=6*E*I/(L^2);
K(4,2)=2*E*I/L;
K(4,3)=-6*E*I/(L^2);
K(4,4)=4*E*I/L;
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% DEFINING THE GLOBAL STIFFNESS MATRIX KT OF THE SYSTEM %
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Terms in the first two rows and first two columns are defined only by the first element.
KT=zeros(2+2*n);
KT(1,1)=K(1,1);
KT(1,2)=K(1,2);
KT(1,3)=K(1,3);
KT(1,4)=K(1,4);
KT(2,1)=K(2,1);
KT(2,2)=K(2,2);
KT(2,3)=K(2,3);
KT(2,4)=K(2,4);
KT(3,1)=K(3,1);
KT(3,2)=K(3,2);
KT(4,1)=K(4,1);
KT(4,2)=K(4,2);
% Defining the terms with contribution from only one element.
for i=1:n-2;
    KT(3+2*i,1+2*i)=K(3,1);
    KT(3+2*i,2+2*i)=K(3,2);
```

```

KT(4+2*i,1+2*i)=K(4,1);
KT(4+2*i,2+2*i)=K(4,2);
% Using the symmetry of the stiffness matrix.
KT(1+2*i,3+2*i)=KT(3+2*i,1+2*i);
KT(2+2*i,3+2*i)=KT(3+2*i,2+2*i);
KT(1+2*i,4+2*i)=KT(4+2*i,1+2*i);
KT(2+2*i,4+2*i)=KT(4+2*i,2+2*i);
end
% Defining the terms with contribution from two elements.
for i=0:n-2;
    KT(3+2*i,3+2*i)=K(3,3)+K(1,1);
    KT(3+2*i,4+2*i)=K(3,4)+K(1,2);
    KT(4+2*i,3+2*i)=K(4,3)+K(2,1);
    KT(4+2*i,4+2*i)=K(4,4)+K(2,2);
end
% The last two rows and columns are defined only by the last element.
KT(2*n-1,1+2*n)=K(1,3);
KT(2*n-1,2+2*n)=K(1,4);
KT(2*n,1+2*n)= K(2,3);
KT(2*n,2+2*n)= K(2,4);
KT(1+2*n,2*n-1)=K(3,1);
KT(1+2*n,2*n)= K(3,2);
KT(1+2*n,1+2*n)=K(3,3);
KT(1+2*n,2+2*n)=K(3,4);
KT(2+2*n,2*n-1)=K(4,1);
KT(2+2*n,2*n)= K(4,2);
KT(2+2*n,1+2*n)=K(4,3);
KT(2+2*n,2+2*n)=K(4,4);

```