AERONAUTICAL SOCIETY OF INDIA

Finite Element Analysis of Beams and Developing a Code to Generate Transformation Matrix for Beams

by

Mohd Afeef Badri

03, July, 2012

Contents

1 Theory involved **2**
- 1.1 Beam definition . . . 2
- 1.2 Types of beams . . . 3
- 1.3 Solving a beam problem . . . 3
- 1.4 Analytical approach to solve beam problem . . . 4
- 1.5 FEM to solve beam problem . . . 5
  - 1.5.1 Basic steps involved in FEM . . . 6
  - 1.5.2 Developing stiffness matrix for a beam element . . . 6

2 A C code to generate beam stiffness matrix **10**
- 2.1 Introduction to C programming . . . 10
- 2.2 C code for generating stiffness matrix . . . 10

3 Code verification **13**
- 3.1 Problem . . . 13
- 3.2 Finite element solution . . . 13

Chapter 1

Theory involved

1.1 Beam definition

A beam is a horizontal structural element that is capable of withstanding load primarily by resisting bending. The bending force induced into the material of the beam as a result of the external loads, own weight, span, and external reactions to these loads is called a bending moment.

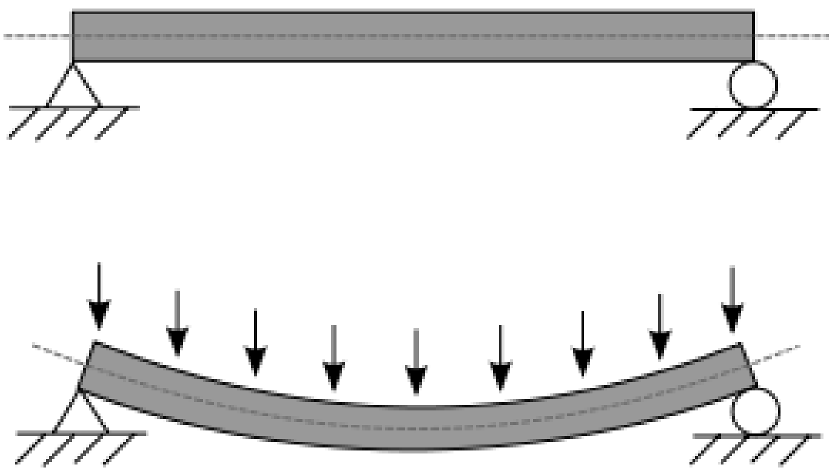

Figure 1.1: a simply supported beam with uniform loading.

Beams are traditionally descriptions of building or civil engineering structural elements, but structures such as truck or automobile frames, machine frames, and other mechanical or structural systems contain beam structures that are designed and analyzed in a similar fashion. It is worth highlighting, beams form an important part of airplane frames. As such are being discussed here in this thesis.

1.2 Types of beams

Beams are characterized by their profile (the shape of their cross-section), their length, and their material. In contemporary construction, beams are typically made of steel, reinforced concrete, or wood. One of the most common types of steel beam is the I-beam or wide-flange beam (also known as a “universal beam” or, for stouter sections, a “universal column”). This is commonly used in steel-frame buildings and bridges. Other common beam profiles are the C-channel, the hollow structural section beam, the pipe, and the angle.

Beams are also described by how they are supported. Supports restrict lateral and/or rotational movements so as to satisfy stability conditions as well as to limit the deformations to a certain allowance. A simple beam is supported by a pin support at one end and a roller support at the other end. A beam with a laterally and rotationally fixed support at one end with no support at the other end is called a cantilever beam. A beam simply supported at two points and having one end or both ends extended beyond the supports is called an overhanging beam.

An important member of airplane wing box is the spar which is an I section beam running generally along the span of the wing. The role of this beam is critical, as it carries not only the load of wing and the fuel around, but also the aerodynamic load exerted by the wing of the aircraft.

1.3 Solving a beam problem

There are two ways that beam bending problems are typically solved: analytically using statics, and computationally using the Finite Element Method (FEM) and a computer simulation. There are inaccuracies associated in both methods. In this project, the beam calculations are performed analytically and using FEM. The results from the two methods are compared and analyzed.

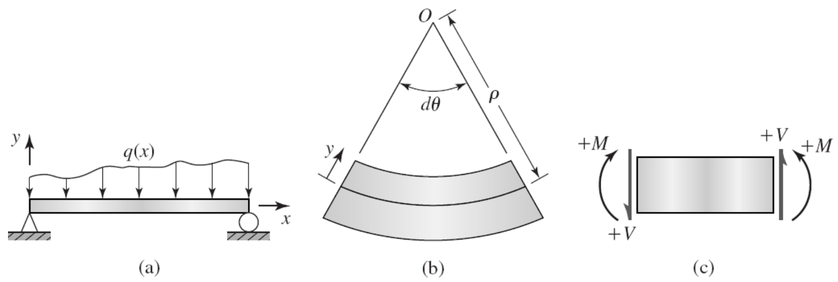

Figure 1.2: (a) simply supported beam subjected to arbitrary (negative) distributed load. (b) Deflected beam element. (c) Sign convention for shear force and bending moment.

1.4 Analytical approach to solve beam problem

Figure 1.2 depicts a simply supported beam subjected to a general, distributed, transverse load $q(x)$ assumed to be expressed in terms of force per unit length. The coordinate system is as shown with x representing the axial coordinate and y the transverse coordinate. The usual assumptions of elementary beam theory are applicable here:

1. the beam is loaded only in the y direction;
2. deflections of the beam are small in comparison to the characteristic dimensions of the beam;
3. the material of the beam is linearly elastic, isotropic, and homogeneous;
4. the beam is prismatic and the cross section has an axis of symmetry in the plane of bending.

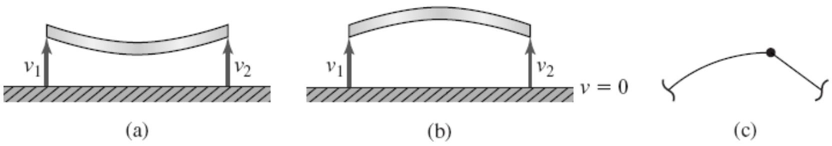

Figure 1.3: (a) and (b) beam elements with identical end deflections but quite different deflection characteristics. (c) Physically unacceptable discontinuity at the connecting node.

Considering a differential length dx of a beam after bending as in fig. 1.3 (with the curvature greatly exaggerated), it is intuitive that the top surface has decreased in length while the bottom surface has increased in length (clearly highlighted in fig. 1.2b). Hence, there is a "layer" within the material that must be undeformed during bending. Refer to fig. 1.2b, assuming that this layer is located distance ρ from the center of curvature O and choosing this layer to correspond to $y = 0$, the length after bending at any position y is expressed as:

$$\mathrm{d}s = (\rho - y)\mathrm{d}\theta. \tag{1.1}$$

And the bending strain is then given by:

$$\varepsilon_x = \frac{\mathrm{d}s - \mathrm{d}x}{\mathrm{d}x} = \frac{(\rho - y)\mathrm{d}\theta - \rho\mathrm{d}\theta}{\rho\mathrm{d}\theta} = -\frac{y}{\rho}. \tag{1.2}$$

From basic calculus, the radius of curvature of a planar curve is given by:

$$\rho = \frac{\left[1 + \left(\frac{\mathrm{d}v}{\mathrm{d}x}\right)^2\right]^{3/2}}{\frac{\mathrm{d}^2 v}{\mathrm{d}x^2}} \tag{1.3}$$

Where $v = v(x)$ represents the deflection curve of the neutral surface. In keeping with small deflection theory, slopes are also small, so this is approximated by:

$$\rho = \frac{1}{\frac{\mathrm{d}^2 v}{\mathrm{d}x^2}} \tag{1.4}$$

Such that the normal strain in the direction of the longitudinal axis as a result of bending is:

$$\varepsilon_x = -y\frac{\mathrm{d}^2 v}{\mathrm{d}x^2} \tag{1.5}$$

And the corresponding normal stress is:

$$\sigma_x = E\varepsilon_x = -Ey\frac{\mathrm{d}^2 v}{\mathrm{d}x^2} \tag{1.6}$$

Where E is the modulus of elasticity of the beam material. Equation (1.6) shows that, at a given cross section, the normal stress varies linearly with distance from the neutral surface. As no net axial force is acting on the beam cross section, the resultant force of the stress distribution given by eq. (1.6) must be zero. Therefore, at any axial position x along the length, we have:

$$F_x = \int_A \sigma_x \ \mathrm{d}A = -\int_A Ey\frac{\mathrm{d}^2 v}{\mathrm{d}x^2} \ \mathrm{d}A = 0 \tag{1.7}$$

Similarly, the internal bending moment at a cross section must be equivalent to the resultant moment of the normal stress distribution, so:

$$M(x) = -\int_A y\sigma_x \ \mathrm{d}A = E\frac{\mathrm{d}^2 v}{\mathrm{d}x^2}\int_A y^2 \ \mathrm{d}A. \tag{1.8}$$

The integral term in this equation represents the moment of inertia of the crossectional area about the z axis, so the bending moment expression becomes:

$$M(x) = EI_z\frac{\mathrm{d}^2 v}{\mathrm{d}x^2} \tag{1.9}$$

Combining eqs. (1.6) and (1.9), we obtain the normal stress equation for beam bending:

$$\sigma_x = -\frac{M(x)y}{I_z} = -yE\frac{\mathrm{d}^2 v}{\mathrm{d}x^2} \tag{1.10}$$

Note that the negative sign in this equation ensures that, when the beam is subjected to positive bending moment per the convention depicted in fig. 1.2c, compressive (negative) and tensile (positive) stress values are obtained correctly depending on the sign of the y location value.

1.5 FEM to solve beam problem

The finite element method is a computational scheme to solve field problems in engineering and science. The technique has very wide application, and has been used on problems involving stress analysis, fluid mechanics, heat transfer, diffusion, vibrations, electrical and magnetic fields, etc. The fundamental concept involves dividing the body under study into a finite number of pieces (sub domains) called elements. Particular assumptions are then made on the variation of the unknown dependent variable(s) across each element using so-called interpolation or approximation functions. This approximated variation is quantified in terms of solution values at special element locations called nodes. Through this discretization process, the method sets up an algebraic system of equations for unknown nodal values which approximate the continuous solution. Because element size, shape and approximating scheme can be varied to suit the problem, the method can accurately simulate solutions to problems of complex geometry and loading and thus this technique has become a very useful and practical tool.

1.5.1 Basic steps involved in FEM

- Domain discretization.
- Select element type (shape and approximation).
- Derive element equations (variational and energy methods)
- Assemble element equations to form global system

  $$[\mathbf{K}]\{U\} = \{F\}$$

  where, $[\mathbf{K}]$ is the stiffness or property matrix, $\{U\}$ is the nodal displacement vector, and $\{F\}$ is the nodal force vector.
- Incorporate boundary and initial conditions.
- Solve assembled system of equations for unknown nodal. displacements and secondary unknowns of stress and strain Values.

1.5.2 Developing stiffness matrix for a beam element

Using the elementary beam theory, the 2D beam element can now be developed with the aid of the first theorem of Castigliano. The assumptions and restrictions underlying the development are the same as those of elementary beam theory with the addition of

1. The element is of length L and has two nodes, one at each end.
2. The element is connected to other elements only at the nodes.
3. Element loading occurs only at the nodes.

The basic premise of finite element formulation is to express the continuously varying field variable in terms of a finite number of values evaluated at element nodes; we note that, for the flexure element, the field variable of interest is the transverse displacement $v(x)$ of the neutral surface away from its straight, undeflected position. As depicted in fig. 1.3a and fig. 1.3b, transverse deflection Of a beam is such that the variation of deflection along the length is not adequately described by displacement of the end points only. The end deflections can be identical, as illustrated, while the deflected shape of the two cases is quite different. Therefore, the flexure element formulation must take into account the Slope (rotation) of the beam as well as end-point displacement. In addition to avoiding the potential ambiguity of displacements, inclusion of beam element nodal rotations ensures compatibility of rotations at nodal connections between elements, thus precluding the physically unacceptable discontinuity depicted in fig. 1.3c.

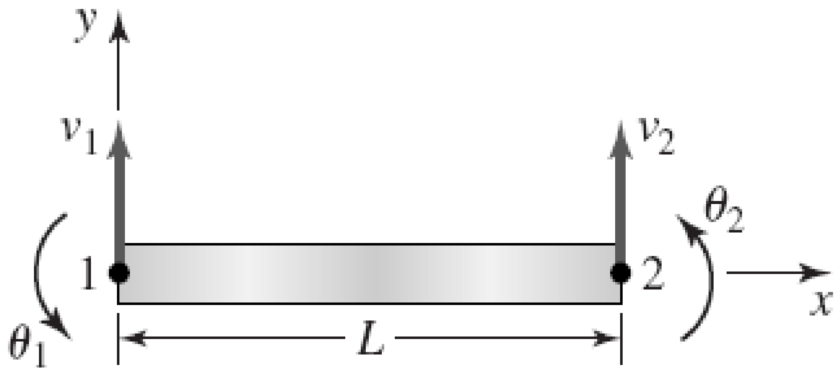

Figure 1.4: beam element nodal displacements shown in a positive sense.

In light of these observations regarding rotations, the nodal variables to be associated with a flexure element are as depicted in fig. 1.4. Element nodes 1 and 2 are located at the ends of the element, and the nodal variables are the transverse displacements v_1 and v_2 at the nodes and the slopes (rotations) θ_1 and θ_2. The nodal variables as shown are in the positive direction, and it is to be noted that the slopes are to be specified in radians. For convenience, the superscript (e) indicating element properties is not used at this point, as it is understood in context that the current discussion applies to a single element. When multiple elements are involved in examples to follow, the superscript notation is restored. The displacement function $v(x)$ is to be discretized such that

$$v(x) = f\left(v_1, v_2, \theta_1, \theta_2, x\right). \tag{1.11}$$

Subject to the boundary conditions

$$\begin{aligned} v\left(x = x_1\right) &= v_1 \\ v\left(x = x_2\right) &= v_2 \\ \left.\frac{\mathrm{d}v}{\mathrm{d}x}\right|_{x=x_1} &= \theta_1 \\ \left.\frac{\mathrm{d}v}{\mathrm{d}x}\right|_{x=x_2} &= \theta_2 \end{aligned} \tag{1.12}$$

Considering the four boundary conditions and the one-dimensional nature of The problem in terms of the independent variable, we assume the displacement function in the form

$$v(x) = a_0 + a_1 x + a_2 x^2 + a_3 x^3 \tag{1.13}$$

Application of the boundary conditions in succession yields

$$\begin{aligned} v(x = 0) &= v_1 = a_0 \\ v(x = L) &= v_2 = a_0 + a_1 L + a_2 L^2 + a_3 L^3 \\ \left.\frac{\mathrm{d}v}{\mathrm{d}x}\right|_{x=0} &= \theta_1 = a_1 \\ \left.\frac{\mathrm{d}v}{\mathrm{d}x}\right|_{x=L} &= \theta_2 = a_1 + 2a_2 L + 3a_3 L^2 \end{aligned} \tag{1.14}$$

The above given equations are solved simultaneously to obtain the coefficients in terms of the nodal variables as

$$\begin{aligned} a_0 &= v_1 \\ a_1 &= \theta_1 \\ a_2 &= \frac{3}{L^2}(v_2 - v_1) - \frac{1}{L}(2\theta_1 + \theta_2) \\ a_3 &= \frac{2}{L^3}(v_1 - v_2) + \frac{1}{L^2}(\theta_1 + \theta_2) \end{aligned} \tag{1.15}$$

Substituting into Equation (1.13) and collecting the coefficients of the nodal variables results in the expression

$$\begin{aligned} v(x) = &\left(1 - \frac{3x^2}{L^2} + \frac{2x^3}{L^3}\right) v_1 + \left(x - \frac{2x^2}{L} + \frac{x^3}{L^2}\right)\theta_1 \\ &+ \left(\frac{3x^2}{L^2} - \frac{2x^3}{L^3}\right) v_2 + \left(\frac{x^3}{L^2} - \frac{x^2}{L}\right)\theta_2 \end{aligned} \tag{1.16}$$

Which is of the form

$$v(x) = N_1(x)v_1 + N_2(x)\theta_1 + N_3(x)v_2 + N_4(x)\theta_2 \tag{1.17}$$

Or, in matrix notation

$$v(x) = \begin{bmatrix} N_1 & N_2 & N_3 & N_4 \end{bmatrix} \begin{Bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{Bmatrix} = [N]\{\delta\} \tag{1.18}$$

Where N_1, N_2, N_3 and N_4 are the interpolation functions that describe the distribution of displacement in terms of nodal values in the nodal displacement vector $\{\delta\}$.

Stress distribution on a cross section located at axial position x is given by:

$$\sigma_x(x, y) = -yE\frac{\mathrm{d}^2[N]}{\mathrm{d}x^2}\{\delta\} \tag{1.19}$$

Since the normal stress varies linearly on a cross section, the maximum and minimum values on any cross section occur at the outer surfaces of the element, Where distance y from the neutral surface is largest. As is customary, we take the maximum stress to be the largest tensile (positive) value and the minimum to be the largest compressive (negative) value. Hence, we rewrite eq. (1.19) as:

$$\begin{aligned} \sigma_x(x) = y_{\max}E \Bigg[&\left(\frac{12x}{L^3} - \frac{6}{L^2}\right) v_1 + \left(\frac{6x}{L^2} - \frac{4}{L}\right)\theta_1 + \left(\frac{6}{L^2} - \frac{12x}{L^3}\right) v_2 \\ &+ \left(\frac{6x}{L^2} - \frac{2}{L}\right)\theta_2 \Bigg] \end{aligned} \tag{1.20}$$

the above equation indicates a linear variation of normal stress along the length of the element and since, once the displacement solution is obtained, the nodal values are known constants, we need calculate only the stress values at the cross sections corresponding to the nodes; that is, at $x = 0$ and $x = L$. The stress values at the nodal sections are given by:

$$\begin{aligned}\sigma_x(x=0) &= y_{\max} E\left[\frac{6}{L^2}(v_2 - v_1) - \frac{2}{L}(2\theta_1 + \theta_2)\right]\\ \sigma_x(x=L) &= y_{\max} E\left[\frac{6}{L^2}(v_1 - v_2) + \frac{2}{L}(2\theta_2 + \theta_1)\right]\end{aligned} \tag{1.21}$$

Writing the strain energy equation of bending for any constant cross-section beam that obeys the assumptions of elementary beam theory.

$$U_e = \frac{EI_z}{2}\int_0^L \left(\frac{\mathrm{d}^2 v}{\mathrm{d}x^2}\right)^2 \mathrm{d}x \tag{1.22}$$

For the strain energy of the finite element being developed, we substitute the discretized displacement relation of eq. (1.13) to obtain

$$U_e = \frac{EI_z}{2}\int_0^L \left(\frac{\mathrm{d}^2 N_1}{\mathrm{d}x^2}v_1 + \frac{\mathrm{d}^2 N_2}{\mathrm{d}x^2}\theta_1 + \frac{\mathrm{d}^2 N_3}{\mathrm{d}x^2}v_2 + \frac{\mathrm{d}^2 N_4}{\mathrm{d}x^2}\theta_2\right)^2 \mathrm{d}x \tag{1.23}$$

Applying the first theorem of Castigliano to the strain energy function with respect to nodal displacement v_1 gives the transverse force at node 1 as

$$\frac{\partial U_e}{\partial v_1} = F_1 = EI_z\int_0^L \left(\frac{\mathrm{d}^2 N_1}{\mathrm{d}x^2}v_1 + \frac{\mathrm{d}^2 N_2}{\mathrm{d}x^2}\theta_1 + \frac{\mathrm{d}^2 N_3}{\mathrm{d}x^2}v_2 + \frac{\mathrm{d}^2 N_4}{\mathrm{d}x^2}\theta_2\right)\frac{\mathrm{d}^2 N_1}{\mathrm{d}x^2}\,\mathrm{d}x \tag{1.24}$$

While application of the theorem with respect to the rotational displacement gives the moment as

$$\frac{\partial U_e}{\partial \theta_1} = M_1 = EI_z\int_0^L \left(\frac{\mathrm{d}^2 N_1}{\mathrm{d}x^2}v_1 + \frac{\mathrm{d}^2 N_2}{\mathrm{d}x^2}\theta_1 + \frac{\mathrm{d}^2 N_3}{\mathrm{d}x^2}v_2 + \frac{\mathrm{d}^2 N_4}{\mathrm{d}x^2}\theta_2\right)\frac{\mathrm{d}^2 N_2}{\mathrm{d}x^2}\,\mathrm{d}x \tag{1.25}$$

Similar results are obtained for node 2. Clubbing together the equations for node 1 and 2 we have

$$\begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14}\\ k_{21} & k_{22} & k_{23} & k_{24}\\ k_{31} & k_{32} & k_{33} & k_{34}\\ k_{41} & k_{42} & k_{43} & k_{44}\end{bmatrix}\begin{Bmatrix} v_1\\ \theta_1\\ v_2\\ \theta_2\end{Bmatrix} = \begin{Bmatrix} F_1\\ M_1\\ F_2\\ M_2\end{Bmatrix} \tag{1.26}$$

Where k_{mn}, are the coefficients of the element stiffness matrix.

$$k_{mn} = k_{nm} = EI_z\int_0^L \frac{\mathrm{d}^2 N_m}{\mathrm{d}x^2}\frac{\mathrm{d}^2 N_n}{\mathrm{d}x^2}\,\mathrm{d}x \quad m,n = 1,4 \tag{1.27}$$

Using the above integral the complete stiffness matrix for the flexure element is then written as

$$[k_e] = \frac{EI_z}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L\\ 6L & 4L^2 & -6L & 2L^2\\ -12 & -6L & 12 & -6L\\ 6L & 2L^2 & -6L & 4L^2\end{bmatrix} \tag{1.28}$$

Chapter 2

A C code to generate beam stiffness matrix

To develop the code for generating stiffness matrix for any beam element C programming language was used. Theory as mentioned in the preceding chapter was used in doing so.

2.1 Introduction to C programming

In computing, C is a general purpose programming Language. Its design provides constructs that map efficiently to typical machine instructions, and therefore it found lasting use in applications that had formerly been coded in assembly language. C is one of the most widely used programming languages of all time. Thus using the language code to generate stiffness matrix can be generated.

2.2 C code for generating stiffness matrix

The following code written in C employs the FEM theory given in the perilous chapter.

```c
#include <stdio.h>
#include <math.h>

main()
{

  int i ,j , m, k , nodes;
  float L[6][1], l[10][1],c[10][1], T[30][30] ,n,q ,E ,A1 ,r,I, EI,
    L1,o;

  printf ("\t \t FEM PROGRAM FOR SIMPLE BEAM ANALYSIS \n \n");

  printf (" Enter the young modulus of material in (N/m^2) E= ");
  scanf  ("%f", &E);

  printf (" Enter the moment of inertia for the section I =");
  scanf ("%f%", &I);

  printf (" Enter the cross-sectional area A =");
```

```
scanf  ("%f", &A1);

EI = E*I;

printf (" Enter the total length of beam ");
scanf ("%f", &L1);

printf (" Enter the number of elements ");
scanf ("%d", &m);

nodes = m+1;

printf ("\n \nSTARTING FROM LEFT START ENTERING THE NODE
  CONDITIONS \n");


for(i=0; i< nodes ;i++)
{
  for(j=0;j<1;j++)
  {
    printf ( "NODE %d load ", i);
    scanf("%f",&l[i][j]);
  }
}

printf ("\n");


for(i=0;i< nodes;i++)
{
  for(j=0;j<1;j++)
  {
    printf ( "NODE %d moment ", i);
    scanf("%f",&c[i][j]);
  }
}

printf( "\nENTER THE LENGTH OF EACH ELEMENT \n");


for(i=0;i< m;i++)
{
  for(j=0;j<1;j++)
  {
    printf ( "NODE %d length ", i);
    scanf("%f",&L[i][j]);
  }
}

T[0][0]=12/((L[0][0])*(L[0][0])*(L[0][0]));
T[0][1]=T[1][0]=T[0][3]=T[3][0]=6/((L[0][0])*(L[0][0]));
T[2][0]=T[0][2]=-12/((L[0][0])*(L[0][0])*(L[0][0]));
T[2][1]=T[1][2]=-6/((L[0][0])*(L[0][0]));
T[3][1]=T[1][3]=2/((L[0][0]));
T[1][1]=4/((L[0][0]));

k=2*nodes;

if (k==4)
{
  T[2][2]=12/((L[0][0])*(L[0][0])*(L[0][0]));
  T[2][3]=T[3][2]= -6/((L[0][0])*(L[0][0]));
```

```
    T[3][3]= 4/(L[0][0]);
  }

  if (k>4)
  {
    for (i=1; i<m ; i++)
    {
      T[i*2][i*2] = (12/ ((L[i][0])*(L[i][0])*(L[i][0]))) + (12/((L
    [(i-1)][0])*(L[(i-1)][0])*(L[(i-1)][0])));

      T[i*2][(i*2)+1] = T[(i*2)+1][i*2] = (6/ ((L[i][0])*(L[i][0]))
    ) - (6/((L[(i-1)][0])*(L[(i-1)][0])));

      T[(i*2)+1][(i*2)+1] = (4/ ((L[i][0]))) + (4/((L[(i-1)][0])));

      T[i*2][(i*2)+2] = T[(i*2)+2][i*2] = -12/((L[i][0])*(L[i][0])
    *(L[i][0]));

      T[i*2][(i*2)+3] = T[(i*2)+3][i*2] =6/((L[i][0])*(L[i][0]));

      T[(i*2)+1][(i*2)+2] = T[(i*2)+2][(i*2)+1] = -6/((L[i][0])*(L[
    i][0]));

      T[(i*2)+1][(i*2)+3] = T[(i*2)+3][(i*2)+1] = 2 / ((L[i][0]));


    }
  }

  T[m*2][m*2] = 12 / ((L[m-1][0])*(L[m-1][0])*(L[m-1][0]));

  T[m*2][(m*2)+1] = T[(m*2)+1][m*2] = -6/((L[(m-1)][0])*(L[(m-1)
    ][0]));

  T[(m*2)+1][(m*2)+1] = 4/ ((L[(i-1)][0]));


  printf ("\nSTIFFNESS MATRIX IS \n");

  for(i=0;i< k;i++)
  {
    printf ("\n");
    for(j=0;j<k;j++)
    {
      printf ( " %.1f  ", T[i][j]);

    }
  }


}
```

Chapter 3

Code verification

In this section a plane beam shown in fig. 3.1, with a two-element FEM discretization. The FEM displacement results will be compared with the exact analytical solution obtained by discontinuity functions.

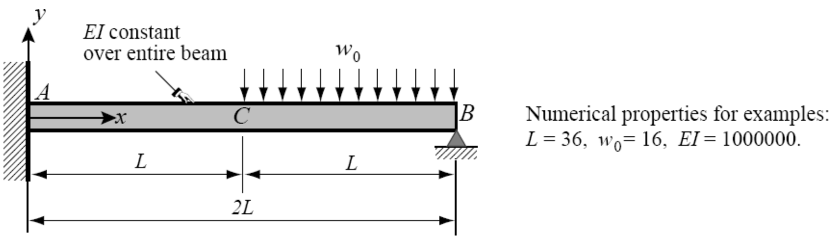

Figure 3.1: a plane beam problem.

3.1 Problem

The beam span is $2L$. It has uniform cross section of elastic modulus E and moment of inertia I about z. The beam is fixed (clamped) at A and simply supported at B. It is loaded by a downward uniform distributed force of magnitude w_0 acting over the right half span $L \leq x \leq 2L$. The problem is statically indeterminate.

3.2 Finite element solution

To illustrate the use of finite elements for beam structures we will discretize the problem of fig. 3.1. Using two plane-beam finite elements as illustrated in fig. 3.2a.

For the problem

- Force $F_1 = 144$ units
- Force $F_2 = -(w_0 + 432 = 144$ units
- Cross-sectional area $A = 16$ units

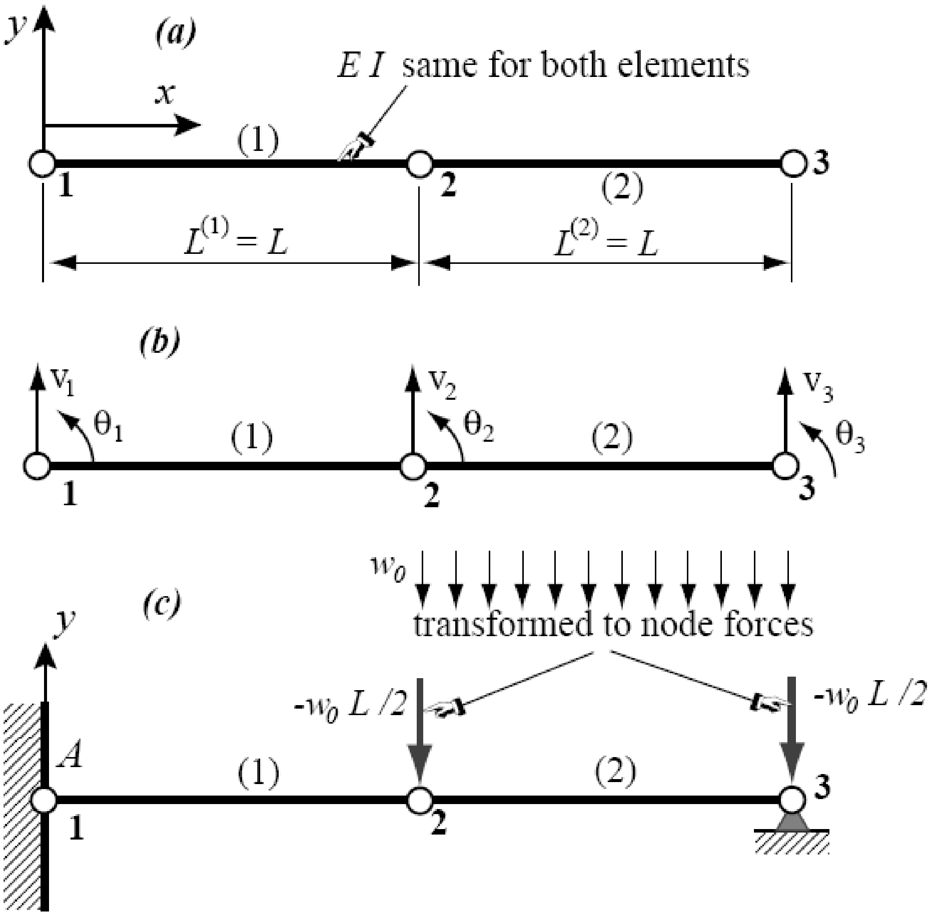

Figure 3.2: (a)two element FEM discretization.(b) The 6 DOF of free free FEM model. (c) Support conditions and applied forces.

Running the code developed to find out the stiffness matrix for this problem we find results presented in fig. 3.3. Hence we obtain

$$\frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L & 0 & 0 \\ 6L & 4L^2 & -6L & 2L^2 & 0 & 0 \\ -12 & -6L & 24 & 0 & -12 & 6L \\ 6L & 2L^2 & 0 & 8L^2 & -6L & 2L^2 \\ 0 & 0 & -12 & -6L & 12 & -6L \\ 0 & 0 & 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}\begin{bmatrix} 0 \\ 0 \\ v_2 \\ \theta_2 \\ 0 \\ \theta_3 \end{bmatrix} = \begin{bmatrix} f_1 \\ m_1 \\ -\frac{1}{2}w_0L \\ 0 \\ f_3 \\ 0 \end{bmatrix} \tag{3.1}$$

this is given by

$$10^6\begin{bmatrix} 0.002058 & 0.018590 & -0.002058 & 0.018590 & 0.000000 & 0.000000 \\ 0.018590 & 0.222222 & -0.018590 & 0.111111 & 0.000000 & 0.000000 \\ -0.002058 & -0.018590 & 0.004115 & 0.000000 & -0.002058 & 0.018590 \\ 0.018590 & 0.111111 & 0.000000 & 0.444444 & -0.018590 & 0.111111 \\ 0.000000 & 0.000000 & -0.002058 & -0.018590 & 0.002058 & -0.018590 \\ 0.000000 & 0.000000 & 0.018590 & 0.111111 & -0.018590 & 0.222222 \end{bmatrix}\begin{bmatrix} 0 \\ 0 \\ v_2 \\ \theta_2 \\ 0 \\ \theta_3 \end{bmatrix} = \begin{bmatrix} f_1 \\ m_1 \\ -\frac{1}{2}w_0L \\ 0 \\ f_3 \\ 0 \end{bmatrix} \tag{3.2}$$

The applied force $f_3 = -1/2w_0L$ becomes part of the reaction taken by the right-end support, since $v_3 = 0$. (In FEM, displacement BCs take precedence

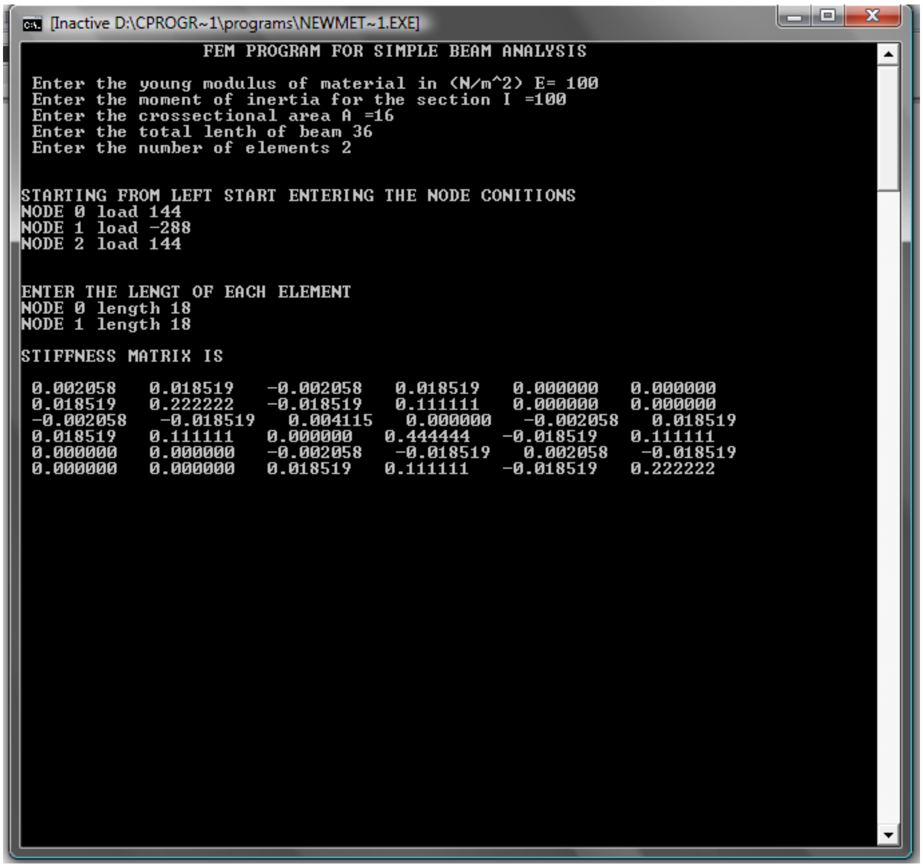

Figure 3.3: stiffness matrix derived from the code.

over force BCs.) Reduce by removing rows and columns 1, 2 and 5, which pertain to the known node displacements:

$$10^6 \begin{bmatrix} 0.004115 & 0.000000 & 0.018519 \\ 0.000000 & 0.444444 & 0.111111 \\ 0.018519 & 0.111111 & 0.222222 \end{bmatrix} \begin{bmatrix} v_2 \\ \theta_2 \\ \theta_3 \end{bmatrix} = \begin{bmatrix} -\frac{1}{2} w_0 L \\ 0 \\ 0 \end{bmatrix} \tag{3.3}$$

Solving the above matrix equation for value of v_2 we get

$$v_2 = v(L) = -0.979776 \tag{3.4}$$

Analytical result of the same problem gives

$$v(L) = -1.3297 \tag{3.5}$$

The difference is approximately 30%. FEM analysis generally produces only approximations to the analytical solution of the mathematical model. The approximation can be improved by using more elements over the beam span

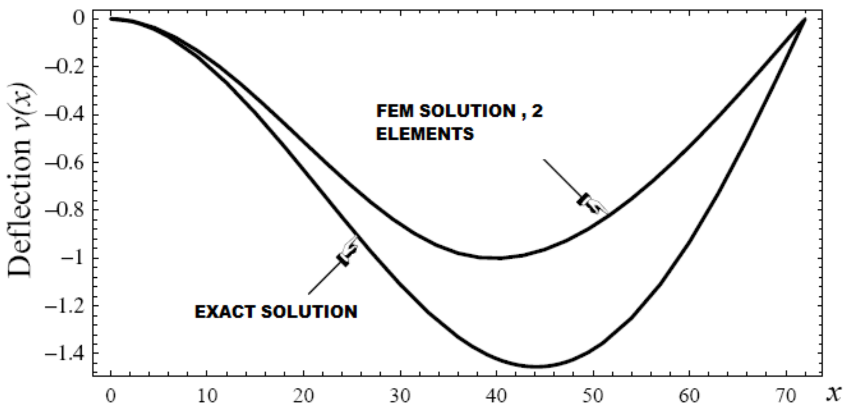

Figure 3.4: deflection of 2 element FEM model versus exact solution.

References

1. C. T. Sun, (2006). Mechanics of aircraft structures. John Wiley & Sons.
2. T. H. G. Megson, (2016). Aircraft structures for engineering students. Butterworth-Heinemann.
3. J. N. Reddy, (1993). An introduction to the finite element method. New York, 27.
4. I. H. Shames, C. L. Dym, & H. Saunders, (1988). Energy and Finite Element Methods in Structural Mechanics.
5. V. Rajaraman, (1995). Computer programming in C. PHI Learning Pvt. Ltd.