CHAPTER

FEM for Beams

5

CHAPTER OUTLINE HEAD

5.1 Introduction

A *beam* is another simple but commonly used structural component. Geometrically, it is usually a slender straight *bar* of an arbitrary cross-section, but it deforms only in directions perpendicular to its axis. Note that the main difference between the beam and the truss is the type of load they carry and the resulting deformation directions. Beams are subjected to transverse loading, including transverse forces and moments that result in transverse deformation. Finite element equations for beams will be developed in this chapter, and the element developed is known as the *beam element*. The basic concepts, procedures, and formulations can also be found in many existing textbooks (see, for example, Petyt, 1990; Reddy, 1993; Rao, 1999; Zienkiewicz and Taylor, 2000).

In beam structures, the beams are joined together by welding (not by pins or hinges, as in the case of truss elements), so that both forces and moments can be transmitted between beams. In this book, the

The Finite Element Method, Second Edition. http://dx.doi.org/10.1016/B978-0-08-098356-1.00005-9

cross-section of the beam structure is assumed to be uniform. If a beam has a varying cross-section, one approach is to divide the beam into a series of shorter beams of constant cross-section to approximate the variation in cross-section. The finite element (FE) matrices for varying cross-sectional area can also be developed with ease using the same concepts that are introduced for a constant cross-section, by allowing the cross-sectional area changing as a function of the axis coordinate. The beam element developed in this chapter is based on the Euler–Bernoulli beam theory that is applicable for thin beams.

5.2 FEM equations

In planar beam elements there are two degrees of freedom (DOF) at a node in its local coordinate system. They are deflection in the y direction, v, and rotation in the $x-y$ plane, θ_z with respect to the z axis (see Section 2.5). Therefore, each two-noded beam element has a total of four DOFs.

5.2.1 Shape function construction

Consider a beam element of length $l=2a$ with nodes 1 and 2 at each end of the element, as shown in Figure 5.1. The local x-axis is taken in the axial direction of the element with its origin at the middle section of the beam. Similar to all other structures, in order to develop the FEM equations, shape functions for the interpolation of the variables from the nodal variables would have to be developed first. As there are four DOFs for a beam element, there should be four shape functions. It is often more convenient if the shape functions are derived from a special set of local coordinates, which is commonly known as the *natural coordinate system*. This natural coordinate system is dimensionless, has its origin at the center of the element, and the element is defined from -1 to $+1$, as shown in Figure 5.1.

The relationship between the natural coordinate system and the local coordinate system can simply be given as the following scaling function

$$\xi = \frac{x}{a} \tag{5.1}$$

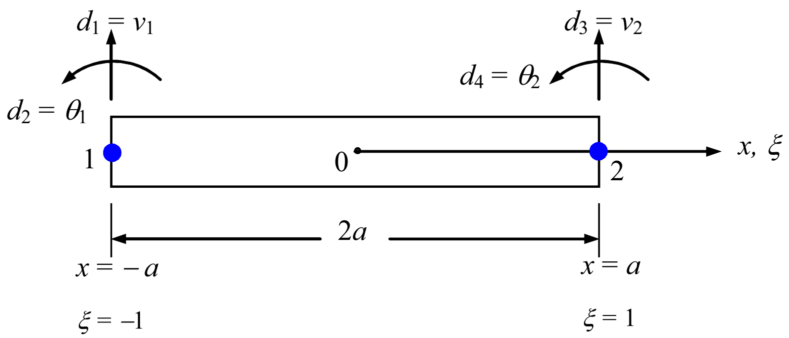

FIGURE 5.1

Beam element and its local coordinate systems: physical coordinates x, and natural coordinates ξ.

To derive the four shape functions in the natural coordinates, the displacement in an element is first assumed in the form of a third order polynomial of ξ that contains four unknown coefficients, a_0 to a_3:

$$v(\xi) = \alpha_0 + \alpha_1 \xi + \alpha_2 \xi^2 + \alpha_3 \xi^3 \tag{5.2}$$

The third order polynomial is chosen here because there are four unknowns in the polynomial, which can be related to the four nodal DOFs in the beam element. The above equation can have the following matrix form:

$$v(\xi) = \underbrace{[1 \;\; \xi \;\; \xi^2 \;\; \xi^3]}_{\mathbf{p}^T(\xi)} \underbrace{\begin{Bmatrix} \alpha_0 \\ \alpha_1 \\ \alpha_2 \\ \alpha_3 \end{Bmatrix}}_{\alpha} \tag{5.3}$$

or

$$v(\xi) = \mathbf{p}^T(\xi)\boldsymbol{\alpha} \tag{5.4}$$

where $\mathbf{p}$ is the vector of basis functions and $\boldsymbol{\alpha}$ is the vector of coefficients, as discussed in Chapters 3 and 4. The rotation θ can be obtained from the differential of Eq. (5.2) using the chain rule, bearing in mind the relationship between x and ξ in Eq. (5.1):

$$\theta = \frac{\partial v}{\partial x} = \frac{\partial v}{\partial \xi}\frac{\partial \xi}{\partial x} = \frac{1}{a}\frac{\partial v}{\partial \xi} = \frac{1}{a}(\alpha_1 + 2\alpha_2 \xi + 3\alpha_3 \xi^2) \tag{5.5}$$

The four unknown coefficients α_0 to α_3 can be determined by utilizing the following four conditions:

At $x = -a$ or $\xi = -1$:

$$\begin{aligned} &(1) \quad v(-1) = v_1 \\ &(2) \quad \left.\frac{\mathrm{d}v}{\mathrm{d}x}\right|_{\xi=-1} = \theta_1 \end{aligned} \tag{5.6}$$

At $x = a$ or $\xi = 1$:

$$\begin{aligned} &(3) \quad v(1) = v_2 \\ &(4) \quad \left.\frac{\mathrm{d}v}{\mathrm{d}x}\right|_{\xi=1} = \theta_2 \end{aligned} \tag{5.7}$$

Substituting Eqs. (5.2) and (5.5) into the above four conditions gives

$$\underbrace{\begin{Bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{Bmatrix}}_{\mathbf{d}_e} = \underbrace{\begin{bmatrix} 1 & -1 & 1 & -1 \\ 0 & 1/a & -2/a & 3/a \\ 1 & 1 & 1 & 1 \\ 0 & 1/a & 3/a & 3/a \end{bmatrix}}_{\mathbf{P}} \underbrace{\begin{Bmatrix} \alpha_0 \\ \alpha_1 \\ \alpha_2 \\ \alpha_3 \end{Bmatrix}}_{\boldsymbol{\alpha}} \tag{5.8}$$

or

$$\mathbf{d}_e = \mathbf{P}\boldsymbol{\alpha} \tag{5.9}$$

Since the moment matrix $\mathbf{P}$ is not singular, solving the above equation for $\boldsymbol{\alpha}$ gives

$$\boldsymbol{\alpha} = \mathbf{P}^{-1}\mathbf{d}_e \tag{5.10}$$

where

$$\mathbf{P}^{-1} = \frac{1}{4}\begin{bmatrix} 2 & a & 2 & -a \\ -3 & -a & 3 & -a \\ 0 & -a & 0 & a \\ 1 & a & -1 & a \end{bmatrix} \tag{5.11}$$

Substituting Eq. (5.10) back into Eq. (5.4) arrives at

$$v = \mathbf{N}(\xi)\mathbf{d}_e \tag{5.12}$$

where $\mathbf{N}$ is a matrix of *shape functions* given by

$$\begin{aligned} \mathbf{N}(\xi) &= \mathbf{p}^T(\xi)\mathbf{P}^{-1} = \underbrace{\begin{bmatrix} 1 & \xi & \xi^2 & \xi^3 \end{bmatrix}}_{\mathbf{p}^T(\xi)} \underbrace{\begin{bmatrix} 2 & a & 2 & -a \\ -3 & -a & 3 & -a \\ 0 & -a & 0 & a \\ 1 & a & -1 & a \end{bmatrix} \frac{1}{4}}_{\mathbf{P}^{-1}} \\ &= [N_1(\xi) \;\; N_2(\xi) \;\; N_3(\xi) \;\; N_4(\xi)] \end{aligned} \tag{5.13}$$

in which the shape functions are found to be

$$\begin{aligned}
N_1(\xi) &= \frac{1}{4}(2 - 3\xi + \xi^3) \\
N_2(\xi) &= \frac{a}{4}(1 - \xi - \xi^2 + \xi^3) \\
N_3(\xi) &= \frac{1}{4}(2 + 3\xi + \xi^3) \\
N_4(\xi) &= \frac{a}{4}(-1 - \xi + \xi^2 + \xi^3)
\end{aligned} \tag{5.14}$$

The first derivatives of this set of shape function becomes,

$$\begin{aligned}
N_1'(\xi) &= \frac{3}{4}(1 + \xi^2) \\
N_2'(\xi) &= \frac{a}{4}(-1 - 2\xi + 3\xi^2) \\
N_3'(\xi) &= \frac{-3}{4}(-1 + \xi^2) \\
N_4'(\xi) &= \frac{a}{4}(-1 + 2\xi + 3\xi^2)
\end{aligned} \tag{5.15}$$

It can be easily confirmed that these shape functions given in Eq. (5.14) satisfy the Delta function property Eq. (3.34), and hence our interpolation using this set of shape functions will be passing node values. The two translational shape functions N_1 and N_3 also satisfy the conditions defined by Eq. (3.41), and hence the correct representation of translational rigid body movements is ensured. However, the two rotational shape functions N_2 and N_4 do not satisfy the conditions of Eq. (3.41). This is because these two shape functions are associated with the rotational DOFs, which are derived from the deflection functions. To ensure the correct representation of the rotational rigid body movement, the condition should be

$$N_2'(\xi = -1) = N_4'(\xi = 1) \tag{5.16}$$

which is also satisfied by Eq. (5.15). Therefore, all rigid body movements are properly represented in the beam element.

5.2.2 Strain matrix

Having now obtained the shape functions, the next step would be to obtain the element strain matrix. Substituting Eq. (5.12) into Eq. (2.47), which gives the relationship between the strain and the deflection, we obtain the normal stress,

$$\varepsilon_{xx} = \mathbf{B}\,\mathbf{d}_e \tag{5.17}$$

where the strain matrix **B** is given by

$$\mathbf{B} = -yL\mathbf{N} = -y\frac{\partial^2}{\partial x^2}\mathbf{N} = -\frac{y}{a^2}\frac{\partial^2}{\partial \xi^2}\mathbf{N} = -\frac{y}{a^2}\mathbf{N}'' \tag{5.18}$$

Note that Eqs. (2.48) and (5.1) have been used for the derivation of the above expression for **B**. Since the expressions for N_i have been derived in Eq. (5.14), we have

$$N'' = [N_1'' \quad N_2'' \quad N_3'' \quad N_4''] \tag{5.19}$$

where

$$\begin{aligned} N_1'' &= \frac{3}{2}\xi,\ N_2'' = \frac{a}{2}(-1+3\xi) \\ N_3'' &= \frac{3}{2}\xi,\ N_4'' = \frac{a}{2}(1+3\xi) \end{aligned} \tag{5.20}$$

5.2.3 Element matrices

With the strain matrix **B** evaluated, we are now ready to obtain the element stiffness and mass matrices. By substituting Eq. (5.18) into Eq. (3.71), the stiffness matrix can be obtained as

$$\begin{aligned} \mathbf{k}_e &= \int_V \mathbf{B}^T \mathbf{c} \mathbf{B}\, \mathrm{d}V = E\underbrace{\int_A y^2 \mathrm{d}A}_{I_z} \int_{-a}^{a} \left(\frac{\partial^2}{\partial x^2}\mathbf{N}\right)^T \left(\frac{\partial^2}{\partial x^2}\mathbf{N}\right) \mathrm{d}x \\ &= EI_z \int_{-1}^{1} \frac{1}{a^4}\left[\frac{\partial^2}{\partial \xi^2}\mathbf{N}\right]^T \left[\frac{\partial^2}{\partial \xi^2}\mathbf{N}\right] a\, \mathrm{d}\xi = \frac{EI_z}{a^3}\int_{-1}^{1} \mathbf{N}''^T N'' \mathrm{d}\xi \end{aligned} \tag{5.21}$$

where $I_z = \int_A y^2\, \mathrm{d}A$ is the second moment of area (or moment of inertia) of the cross-section of the beam with respect to the z axis. Substituting Eq. (5.19) into (5.21), we obtain

$$\mathbf{k}_e = \frac{EI_z}{a^3}\int_{-1}^{1} \begin{bmatrix} N_1''N_1'' & N_1''N_2'' & N_1''N_3'' & N_1''N_4'' \\ N_2''N_1'' & N_2''N_2'' & N_2''N_3'' & N_2''N_4'' \\ N_3''N_1'' & N_3''N_2'' & N_3''N_3'' & N_3''N_4'' \\ N_4''N_1'' & N_4''N_2'' & N_4''N_3'' & N_4''N_4'' \end{bmatrix} \mathrm{d}\,\xi \tag{5.22}$$

Evaluating the integrals in the above equation leads to

$$\mathbf{k}_e = \frac{EI_z}{2a^3}\begin{bmatrix} 3 & 3a & -3 & 3a \\ & 4a^2 & -3a & 2a^2 \\ & & 3 & -3a \\ sy. & & & 4a^2 \end{bmatrix} \tag{5.23}$$

For the mass matrix, we substitute Eq. (5.13) into Eq. (3.75):

$$\begin{aligned} \mathbf{m}_e &= \int_V \rho \mathbf{N}^T \mathbf{N}\, \mathrm{d}V = \rho \int_A \mathbf{N}^T \mathbf{N}\, \mathrm{d}x = \rho A \int_{-1}^{1} \mathbf{N}^T \mathbf{N} a\, \mathrm{d}\xi \\ &= \rho A a \int_{-1}^{1} \begin{bmatrix} N_1N_1 & N_1N_2 & N_1N_3 & N_1N_4 \\ N_2N_1 & N_2N_2 & N_2N_3 & N_2N_4 \\ N_3N_1 & N_3N_2 & N_3N_3 & N_3N_4 \\ N_4N_1 & N_4N_2 & N_4N_3 & N_4N_4 \end{bmatrix} \mathrm{d}\,\xi \end{aligned} \tag{5.24}$$

where A is the area of the cross-section of the beam. Evaluating the integral in the above equation leads to

$$\mathbf{m}_e = \frac{\rho A a}{105}\begin{bmatrix} 78 & 22a & 27 & -13a \\ & 8a^2 & 13a & -6a^2 \\ & & 78 & -22a \\ sy. & & & 8a^2 \end{bmatrix} \tag{5.25}$$

The other necessary element matrix would be the force vector. The nodal force vector for beam elements can again be obtained using the general expressions given in Eqs. (3.78), (3.79), and (3.81). We now assume that the element is loaded by an external, uniformly distributed force f_y along the x-axis, two concentrated forces F_{s1} and F_{s2}, and concentrated moments M_{s1} and M_{s2}, respectively, at nodes 1 and 2; the total nodal force vector can be obtained as

$$\begin{aligned} \mathbf{f}_e &= \int_V \mathbf{N}^T f_b\, \mathrm{d}V + \int_{S_f} \mathbf{N}^T f_s\, \mathrm{d}S_f \\ &= \int_{-a}^{a} A f_b \mathbf{N}^T\, \mathrm{d}x + \int_{S_f} \mathbf{N}^T f_s\, \mathrm{d}S_f \\ &= f_y\, a \int_{-1}^{1} \begin{bmatrix} N_1 \\ N_2 \\ N_3 \\ N_4 \end{bmatrix} \mathrm{d}\xi + \begin{Bmatrix} F_{s1} \\ M_{s1} \\ F_{s2} \\ M_{s2} \end{Bmatrix} = \begin{Bmatrix} f_y a + F_{s1} \\ f_y a^2/3 + M_{s1} \\ f_y a + F_{s2} \\ -f_y a^2/3 + M_{s2} \end{Bmatrix} \end{aligned} \tag{5.26}$$

Note that $f_y = Af_b$ and $dx = a d\xi$ when deriving the above expression. The final FEM equation for a beam element has the form of Eq. (3.89), and the element matrices are defined by Eqs. (5.21), (5.24), and (5.26).

5.3 Remarks

Theoretically, coordinate transformation can also be used to transform the beam element matrices from the local coordinate system into a global coordinate system. However, the transformation is necessary only if there is more than one beam element in the beam structure, and of these there are at least two beam elements of different orientations. A beam structure with at least two beam elements of different orientations is commonly termed a *frame* or *framework*. To analyze frames, frame elements, which carry both axial and bending forces, have to be used, and coordinate transformation is generally required. The formulation for frames is discussed in the next chapter.

5.4 Worked examples

EXAMPLE 5.1

A uniform cantilever beam subjected to a downward force

Consider the cantilever beam as shown in Figure 5.2. The beam is fixed at one end, and it has a uniform cross-sectional area as shown. The beam undergoes static deflection by a downward load of $P = 1000$ N applied at the free end. The dimensions of the beam are shown in the figure, and the beam is made of aluminum whose properties are shown in Table 5.1.

To emphasize the steps involved in solving this simple example, we just use one beam element to solve the deflection. The beam element would have a DOF as shown in Figure 5.1.

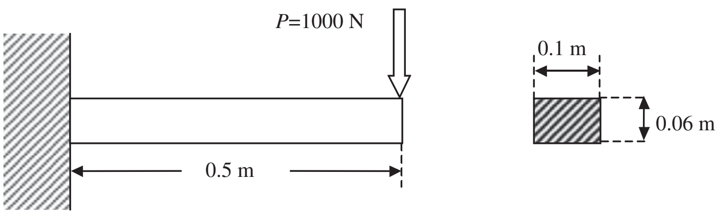

FIGURE 5.2

Cantilever beam of rectangular cross-section, subject to astatic load.

Table 5.1 Material properties of aluminum.

Young's Modulus, E (GPa)	Poisson's Ratio, υ
69.0	0.33

Step 1: Obtaining the element matrices

The first step in formulating the FE equations is to form the element matrices and, in this case, being the only element used, the element matrices are actually the global FE matrices, i.e., no assembly is required. The shape functions for the four DOFs are given in Eq. (5.14).

The element stiffness matrix can be obtained using Eq. (5.21). Note that since this is a static problem, the mass matrix is not required here. The second moment of area of the cross-sectional area about the z axis can be given as

$$I_z = \frac{1}{12}bh^3 = \frac{1}{12}(0.1)\ (0.06)^3 = 1.8 \times 10^{-6}\ \mathrm{m}^4 \tag{5.27}$$

Since only one element is used, the final stiffness matrix of the structure is thus the same as the element stiffness matrix:

$$\begin{aligned}\mathbf{K} = \mathbf{k}_e &= \frac{(69 \times 10^9)\ (1.8 \times 10^{-6})}{2 \times 0.25^3}\begin{bmatrix} 3 & 0.75 & -3 & 0.75 \\ 0.75 & 0.25 & -0.75 & 0.125 \\ -3 & -0.75 & 3 & -0.75 \\ 0.75 & 0.125 & -0.75 & 0.25 \end{bmatrix} \\ &= 3.974 \times 10^6 \begin{bmatrix} 3 & 0.75 & -3 & 0.75 \\ 0.75 & 0.25 & -0.75 & 0.125 \\ -3 & -0.75 & 3 & -0.75 \\ 0.75 & 0.125 & -0.75 & 0.25 \end{bmatrix}\end{aligned} \tag{5.28}$$

The FE equation becomes

$$3.974 \times 10^6 \begin{bmatrix} 3 & 0.75 & -3 & 0.75 \\ 0.75 & 0.25 & -0.75 & 0.125 \\ -3 & -0.75 & 3 & -0.75 \\ 0.75 & 0.125 & -0.75 & 0.25 \end{bmatrix} \begin{Bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} Q_1 \\ M_1 \\ Q_2 = P \\ M_2 = 0 \end{Bmatrix} \tag{5.29}$$

Note that, at node 1, the beam is clamped. Therefore, the shear force and moment at this node should be the reaction force and moment, which are unknowns before the FEM equation is solved for the displacements. To solve Eq. (5.29), we need to impose the displacement boundary condition at the clamped node.

Step 2: Applying boundary conditions

The beam is fixed or clamped at one end. This implies that at that end, the deflection, v_1, and the slope, θ_1, are both equal to zero:

$$v_1 = \theta_1 = 0 \tag{5.30}$$

The imposition of the above displacement boundary condition leads to the removal of the first and second rows and columns of the stiffness matrix:

$$3.974\times 10^{6}\begin{bmatrix} \cancel{3} & \cancel{0.75} & \cancel{-3} & \cancel{0.75} \\ \cancel{0.75} & \cancel{0.25} & \cancel{-0.75} & \cancel{0.125} \\ \cancel{-3} & \cancel{-0.75} & 3 & -0.75 \\ \cancel{0.75} & \cancel{0.125} & -0.75 & 0.25 \end{bmatrix}\begin{Bmatrix} \cancel{\upsilon_1 = 0} \\ \cancel{\theta_1 = 0} \\ \upsilon_2 \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} \cancel{Q_1} \\ \cancel{M_1} \\ Q_2 = P \\ M_2 = 0 \end{Bmatrix} \tag{5.31}$$

The reduced stiffness matrix becomes a 2×2 matrix of

$$\mathbf{K} = 3.974 \times 10^{6}\begin{bmatrix} 3 & -0.75 \\ -0.75 & 0.25 \end{bmatrix} \tag{5.32}$$

The FE equation, after the imposition of the displacement condition, becomes

$$\mathbf{Kd} = \mathbf{F} \tag{5.33}$$

where

$$\mathbf{d}^T = [v_2 \;\; \theta_2] \tag{5.34}$$

and the force vector $\mathbf{F}$ is given as

$$\mathbf{F} = \begin{Bmatrix} -1000 \\ 0 \end{Bmatrix} \text{N} \tag{5.35}$$

Note that, although we do not know the reaction shear force Q_1 and the moment M_1, it does not prevent it from solving the FEM equation because we know υ_1 and θ_1 instead. This allows us to remove the unknowns of Q_1 and M_1 from the original FEM equation. We will come back to calculate the unknowns of Q_1 and M_1 after we have solved the FEM equations for all the displacements (deflections and rotations).

Step 3: Solving the FE matrix equation

The last step in this simple example would be to solve Eq. (5.33) to obtain υ_2 and θ_2. In this case, Eq. (5.33) is actually two simultaneous equations involving two unknowns, and can be easily solved manually. Of course, when we have more unknowns or DOFs, some numerical methods of solving the matrix equation might be required. The solution to Eq. (5.33) is

$$\begin{aligned} v_2 &= -3.3548 \times 10^{-4} \text{ m} \\ \theta_2 &= -1.0064 \times 10^{-3} \text{ rad} \end{aligned} \tag{5.36}$$

After v_2 and θ_2 have been obtained, they can be substituted back into the first two equations of Eq. (5.29) to obtain the reaction shear force at node 1:

$$
\begin{aligned}
Q_1 &= 3.974 \times 10^6 \ (-3v_2 + 0.75\theta_2) \\
&= 3.974 \times 10^6[-3 \times (-3.335 \times 10^{-4}) + 0.75 \times (-1.007 \times 10^{-3})] \\
&= 1000.12 \text{ N}
\end{aligned} \tag{5.37}
$$

and the reaction moment at node 1:

$$
\begin{aligned}
M_1 &= 3.974 \times 10^6 \ (-0.75v_2 + 0.125\theta_2) \\
&= 3.974 \times 10^6[-0.75 \times (-3.335 \times 10^{-4}) + 0.125 \times (-1.007 \times 10^{-3})] \\
&= 500.02 \text{ Nm}
\end{aligned} \tag{5.38}
$$

This completes the solution process of this problem.

Note that this solution is exactly the same as the analytical solution in terms of the deflection. We again observe the reproduction feature of the FEM that was revealed in Example 4.1. In this case, it is because the exact solution of the deflection for the cantilever thin beam is a third order polynomial, which can be obtained easily by solving the strong form of the system equation of beam given by Eq. (2.59) with $f_y = 0$. On the other hand, the shape functions used in our FEM analysis are also third order polynomials (see Eq. (5.14) or Eq. (5.2)). Therefore, the exact solution of the problem is included in the set of assumed deflections. The FEM based on Hamilton's principle has indeed reproduced the exact solution. This is, of course, also true if we were to calculate the deflection at anywhere else other than the nodes. For example, to compute the deflection at the center of the beam, we can use Eq. (5.12) with $x = 0$, or in the natural coordinate system, $\xi = 0$, and substituting the values calculated at the nodes:

$$
v_{\xi=0} = \mathbf{N}_{\xi=0}\mathbf{d}_e = \begin{bmatrix} \frac{1}{2} & \frac{1}{16} & \frac{1}{2} & -\frac{1}{16} \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ -3.355 \times 10^{-4} \\ -0.007 \times 10^{-3} \end{Bmatrix} = -1.048 \times 10^{-4} \text{m} \tag{5.39}
$$

To calculate the rotation at the center of the beam, the derivatives of the shape functions are used as follows:

$$
\begin{aligned}
\theta_{\xi=0} &= \left(\frac{\mathrm{d}v}{\mathrm{d}x}\right)_{\xi=0} = \left(\frac{\mathrm{d}\mathbf{N}}{\mathrm{d}x}\right)_{\xi=0} \mathbf{d}_e = \begin{bmatrix} -3 & -\frac{1}{4} & 3 & -\frac{1}{4} \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ -3.355 \times 10^{-4} \\ -0.007 \times 10^{-3} \end{Bmatrix} \\
&= -7.548 \times 10^{-4} \text{ rad}
\end{aligned} \tag{5.40}
$$

Note that in obtaining $\mathrm{d}\mathbf{N}/\mathrm{d}x$ above, the chain rule of differentiation is used, together with the relationship between x and ξ, as depicted in Eq. (5.1).

5.5 Case study: resonant frequencies of micro-resonant transducer

Making machines as small as insects, or even smaller, has been a dream of scientists for many years. Made possible by present lithographic techniques, such micro-systems are now being produced and applied in our daily lives. Such machines are called micro-electro-mechanical systems (MEMS), usually composed of mechanical and electrical devices. There are many MEMS devices being designed and manufactured today, from micro-actuators and sensors to micro-fluidic devices. The technology has very wide applications in areas such as communication, medical, aerospace, and robotics.

One of the most common MEMS devices is the resonant transducer. Resonant transducers convert externally induced beam strain into a beam resonant frequency change. This change in resonant frequency is then typically detected by implanted piezoresistors or optical techniques. Such resonant transducers are used for the measurement of pressure, acceleration, strain, and vibration. Figure 5.3 shows a micrograph of a micro-polysilicon resonant micro-beam transducer.

Figure 5.3 shows an overall view of the transducer, but the principle of the resonant transducer actually lies in the clamped–clamped bridge on top of a membrane. This bridge is actually located at the center of the micrograph. Figure 5.4 shows a schematic side view of the bridge structure. The resonant frequency of the bridge is related to the force applied to it (between anchor points), its material properties, cross-sectional area, and length. When the membrane deforms, for example, due to a change in pressure, the force applied to the bridge also changes, resulting in a change in the resonant frequency of the bridge.

It is thus important to analyze the resonant frequency of this bridge structure in the design of the resonant transducer. This case study first demonstrates the use of the beam element in the software ABAQUS to solve for the first three resonant frequencies of the bridge. Subsequently, results obtained using ANSYS for the same problem will also be shown for a comparison. More detailed discussions on using commercial software package are given in Chapter 13. The dimensions of the clamped–clamped bridge structure

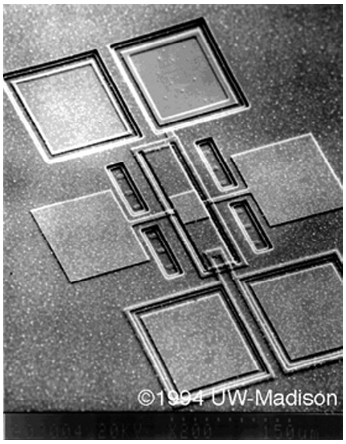

FIGURE 5.3

Resonant micro-beam strain transducer.

Courtesy of Professor Henry Guckel and the University of Wisconsin-Madison.

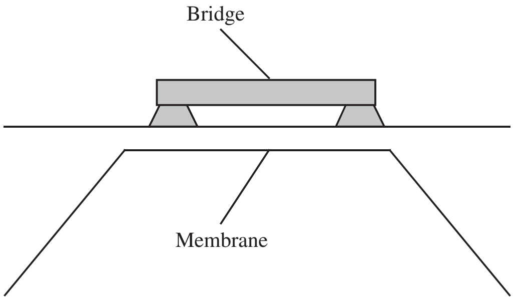

FIGURE 5.4

Bridge in a micro-resonant transducer.

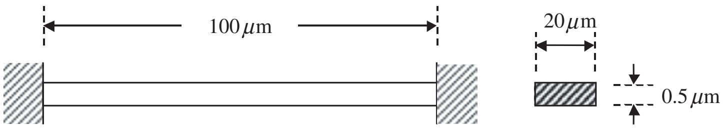

FIGURE 5.5

Geometrical dimensions of clamped–clamped bridge.

Table 5.2 Elastic properties of polysilicon.

Elastic Properties of Polysilicon	
Young's modulus, E	169 GPa
Poisson's ratio, υ	0.262
Density, ρ	2300 kgm^{-3}

shown in Figure 5.5 are used to model a bridge in a micro-resonant transducer. The material properties of polysilicon, of which the resonant transducer is normally made, are shown in Table 5.2.

5.5.1 Modeling

The modeling of the bridge is done using one-dimensional beam elements developed in this chapter. The beam is assumed to be clamped at two ends of the beam. The meshing of the structure should not pose any difficulty, but what is important here is the choice of how many elements to use to give sufficient accuracy. Because the exact solution of free vibration modes of the beam is no longer of a polynomial type, the FEM will not be able to produce the exact solution, but an approximated solution. One naturally becomes concerned with whether the results converge and whether they are accurate.

To start, the beam is meshed uniformly into ten two-nodal beam elements. This simple mesh will serve to show clearly the steps used in ABAQUS. Refined uniform meshes of 20, 40, and 60 elements will then be used to check the accuracy of the results obtained. This is a simplified way of performing what is commonly known as a convergence test. Remember that usually the greater the number of elements, the greater the accuracy. However, we cannot simply use as many elements as possible all the time, since there is usually a limit to the computer resources available. Hence, convergence tests are carried out to determine the optimum number of elements or nodes to be used for a certain problem. By "optimum," we mean the least number of elements or nodes to yield a desired accuracy within the acceptable tolerance.

5.5.2 ABAQUS input file

When using ABAQUS (or any other FEM software package), one often uses a graphic user interface, and almost everything can be done interactively on the screen. The analyst may not even need to know the input file for ABAQUS. This "black-box" setting is useful for beginners. However, understanding the data structure of the input file helps a lot in the appreciation of how the analysis is prepared for the computer to get the job done properly. Once getting used to the input file, the analyst can directly edit the file (in an "old" fashion) and have things (changing settings, modifying models, creating more running cases, etc.) done much more efficiently. Most of all, the analyst can have an important feeling of reassurance on what is being done by the computer. For this reason, we provide this section.

For ABAQUS, the pre-processor generates an input file in the form of a text file that includes information and parameters about the model. The ABAQUS input file for the above described FE model is shown below. In the early days, the analyst had to write these cards manually, but now it is generated by the pre-processors of FEM packages. Understanding the input file aids is important to the modeler in understanding the FEM and to effectively use the FEM packages. The ABAQUS input file for this problem is shown below. Note that the text boxes to the right of the input file are not part of the input file, but explain what the sections of the file mean.

```
* HEADING, SPARSE
ABAQUS job to calculate eigenvalues of beam
**
* NODE
1, 0., 0.
2, 10., 0.
3, 20., 0.
4, 30., 0.
5, 40., 0.
6, 50., 0.
7, 60., 0.
8, 70., 0.
9, 80., 0.
10, 90., 0.
11, 100., 0.
**
**
```

Nodal cards
These define the coordinates of the nodes in the model. The first entry is the node ID, while the second and third entries are the x and y coordinates of the position of the node, respectively.

```
* ELEMENT, TYPE=B23, ELSET=BEAM
1, 1, 2
2, 2, 3
3, 3, 4
4, 4, 5
5, 5, 6
6, 6, 7
7, 7, 8
8, 8, 9
9, 9, 10
10, 10, 11
**
** beam
**
* BEAM SECTION, ELSET=BEAM, SECTION=RECT, MATERIAL=POLYSILI
20., 0.5,
0.0, 0.0, –1.0
**
**
**
**
**
**
**
** polysilicon
**
* MATERIAL, NAME=POLYSILI
**
* DENSITY
2.3E-15,
**
* ELASTIC, TYPE=ISO
169000., 0.262
**
**
* BOUNDARY, OP=NEW
1, 1,, 0.
1, 2,, 0.
2, 1,, 0.
3, 1,, 0.
4, 1,, 0.
5, 1,, 0.
6, 1,, 0.
```

Element (connectivity) cards
These define the element type and what nodes make up the element. B23 represents that it is a planar, cubic, Euler–Bernoulli beam element. There are many other beam element types in the ABAQUS element library. The "ELSET = BEAM" statement is simply for naming this set of elements so that it can be referenced when defining the material properties. In the subsequent data entry, the first entry is the element ID, and the following two entries are the nodes making up the element.

Property cards
These define properties to the elements of set "BEAM." "SECT = RECT" describes the cross-section as a rectangle. ABAQUS provides a choice of other cross-sections. The first data line under "BEAM SECTION" defines the geometry of the cross-section. The second data line defines the normal, which in this case is for a planar beam. It will have the material properties defined under "POLYSILI."

Material cards
These define material properties under the name "POLYSILI." Density and elastic properties are defined. "TYPE=ISO" represents isotropic properties.

Boundary (BC) cards
These define boundary conditions. For nodes 1 and 11, the DOFs 1, 2, and 6 are constrained. For the rest, DOF 1 is constrained. Note that in ABAQUS, a planar beam has x and y translational displacements, as well as rotation about the z axis.

```
7, 1,, 0.
8, 1,, 0.
9, 1,, 0.
10, 1,, 0.
11, 1,, 0.
11, 2,, 0.
**
*BOUNDARY, OP=NEW
I, 6,, 0.
II, 6,, 0.
**
** Step 1, eigen
** LoadCase, Default
**
* STEP, NLGEOM
This load case is the default load case that always appears
*FREQUENCY
3, 0.,,, 30
**
**
**
* NODE PRINT, FREQ=1
U,
* NODE FILE, FREQ=1
U,
**
**
**
* END STEP
```

> *Control cards*
> These indicate the analysis step. In this case it is a "FREQUENCY" analysis or an eigenvalue analysis.

> *Output control cards*
> These define the output required. For example, in this case, we require the nodal output, displacement "U."

The input file above shows how a basic ABAQUS input file is set up. Note that all the input file does is provide the information necessary so that the program can utilize them to formulate and solve the FE equations. It may also be noticed that in the input file, there is no mention of the units of measurement used. This implies that the units must be consistent throughout the input file in all the information provided. For example, if the coordinate values of the nodes are in micrometers, the units for other values like the Young's modulus, density, forces, and so on must also undergo the necessary conversions in order to be consistent, before they are keyed into the pre-processor of ABAQUS and hence the input file. It is noted that in this case study, all the units are converted into micrometers to be consistent with the geometrical dimensions, as can be seen from the values of Young's modulus and density. This is the case for most FE software, and many times, errors in analysis occur due to negligence in ensuring the units' consistency. More details regarding ABAQUS input files will be provided in Chapter 13.

5.5.3 Solution process

Let's now try to relate the information provided in the input file with what is formulated in this chapter. The first part of the ABAQUS input normally describes the nodes and their coordinates (position). These lines are often called "nodal cards."[1] The second part of the input file are the so-called "element cards." Information regarding the definition of the elements using nodes is provided. For example, element 1 is formed by nodes 1 and 2. The element cards give the connectivity of the element or the order of the nodal number that forms the element. The connectivity is very important because a change in the order of the nodal numbers may lead to a breakdown of the computation. The connectivity is also used as the index for the direct assembly of the global matrices (see Example 4.2). This element and nodal information is required for determining the stiffness matrix (Eq. (5.21)) and the mass matrix (Eq. (5.24)).

The property cards define the properties (type of element, cross-sectional property, etc.) of the elements, as well as the material the element is made of. The cross-section of the element is defined here as it is required for computation of the moment of area about the z axis, which is in turn used in the stiffness matrix. The material properties defined are also a necessity for the computation of both the stiffness (elastic properties) and mass matrices (density).

The boundary cards (BC cards) define the boundary conditions for the model. In ABAQUS, a node of a *general beam element* (equivalent to the frame element, to be discussed in the next chapter) in the XY plane has three DOFs: Translational displacements in the x and y directions (1, 2), and the rotation about the z axis (6). To model just the transverse displacements and rotation as depicted in the formulation in this chapter, the x-displacement DOFs are constrained here. Hence it can be seen from the input file that the DOF "1" is constrained for all nodes. In addition to this, the two nodes at the ends, nodes 1 and 11, also have their "2" and "6" DOFs constrained to simulate clamped ends. Just as in the worked example previously, constraining these DOFs would effectively reduce the dimension of the matrix.

We should usually also have *load cards*. Because this case study is an eigenvalue analysis, there are no external loadings, and hence there is no need to define any loadings in the input file.

The control cards are used to control the analysis, such as defining the type of analysis required. ABAQUS uses the subspace iteration scheme by default to evaluate the eigenvalues of the equation of motion. This method is a very effective method of determining a number of lowest eigenvalues and corresponding eigenvectors for a very large system of several thousand DOFs. The procedure is as follows:

i. To determine the n lowest eigenvalues and eigenvectors, select a starting matrix $\mathbf{X}_1$ having m $(> n)$ columns.

ii. Solve the equation $\mathbf{K}\overline{\mathbf{X}}_{k+1} = \mathbf{M}\mathbf{X}_k$ for $\overline{\mathbf{X}}_{k+1}$

iii. Calculate $\mathbf{K}_{k+1} = \overline{\mathbf{X}}_{k+1}^T\mathbf{K}\overline{\mathbf{X}}_{k+1}$ and $\mathbf{M}_{k+1} = \overline{\mathbf{X}}_{k+1}^T\,\mathbf{M}\overline{\mathbf{X}}_{k+1}$, where the dimension of $\mathbf{K}_{k+1}$ and $\mathbf{M}_{k+1}$ are of m by m.

iv. Solve the reduced eigenvalue problem $\mathbf{K}_{k+1}\psi_{k+1} - \mathbf{M}\psi_{k+1}\Lambda_{k+1} = 0$ for m eigenvalues, which are the diagonal terms of the diagonal matrix Λ_{k+1}, and for eigenvectors ψ_{k+1}.

[1] In the early 1980s, the input files were recorded on paper cards with punched holes, where each card recorded one line of letters or numbers represented by the arrangement of the holes. Optical devices were then used to read-in these cards for the computer. We do not use such "cards" anymore, but the name is still widely used.

v. Calculate the improved approximation to the eigenvectors of the original system using $\mathbf{X}_{k+1} = \overline{\mathbf{X}}_{k+1}\ \Psi_{k+1}$.

vi. Repeat the process until the eigenvalues and eigenvectors converge to the lowest eigenvectors to desired accuracy.

By specifying the line "*FREQUENCY" in the analysis step, ABAQUS will carry out a similar algorithm to that briefly explained above. The line after the "*FREQUENCY" contains some data which ABAQUS uses to aid the procedure. The first entry refers to the number of eigenvalues required (in this case, 3). The second refers to the maximum frequency of interest. This will limit the frequency range, and therefore anything beyond this frequency will not be calculated. In this case, no maximum frequency range will be specified. The third is to specify shift points, which are used to ensure that the stiffness matrix is not singular. Here, again, it is left blank since it is not necessary. The fourth is the number of columns of the starting matrix $\mathbf{X}_1$ to be used. It is left blank again, and thus ABAQUS will use its default setting. The last entry is the number of iterations, which in this case is 30.

Output control cards are used for selecting the data that needs to be output. This is very useful for large scale computation that produces huge data files; one needs to limit the output to what is really needed.

Once the input file is created, one can then invoke ABAQUS to execute the analysis, and the results will be written into an output file that can be read by the post-processor.

5.5.4 Results and discussion

Using the above input file, an analysis to calculate the eigenvalues, and hence the natural resonant frequencies of the bridge structure, is carried out using ABAQUS. Other than the 10-element mesh as shown in Figure 5.6, which is also depicted in the input file, a simple convergence test is carried out. Hence, there are similar uniform meshes using 20, 40, and 60 elements. All the frequencies obtained are given in Table 5.3. Because the clamped–clamped beam structure is a simple problem, it is possible to evaluate the natural frequencies analytically. The results obtained from analytical calculations are also shown in Table 5.3 for comparison.

From the table, it can be seen that the finite element results give very good approximations compared to the analytical results. Even with just 10 elements, the error of mode 1 frequency is about 0.016% from the analytical calculations. It can also be seen that as the number of elements increases, the finite element results get closer and closer to the analytical calculations, and converge such that the results obtained for 40 and 60 elements show no difference up to the fourth decimal place. What this implies is that in finite element analyses, the finer the mesh or the greater the number of elements used, the more accurate the results. However, using more elements will use up more computer resources and

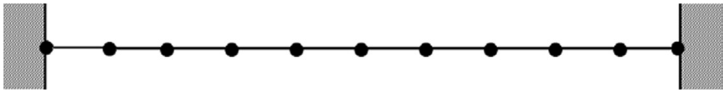

FIGURE 5.6

Ten element mesh of clamped-clamped bridge.

Table 5.3 Resonant frequencies of bridge using FEA (ABAQUS) and analytical calculations.

Number of 2-Node Beam Elements	Natural Frequency (Hz)		
	Mode 1	Mode 2	Mode 3
10	4.4058×10^5	1.2148×10^6	2.3832×10^6
20	4.4057×10^5	1.2145×10^6	2.3809×10^6
40	4.4056×10^5	1.2144×10^6	2.3808×10^6
60	4.4056×10^5	1.2144×10^6	2.3808×10^6
Analytical calculations	4.4051×10^5	1.2143×10^6	2.3805×10^6

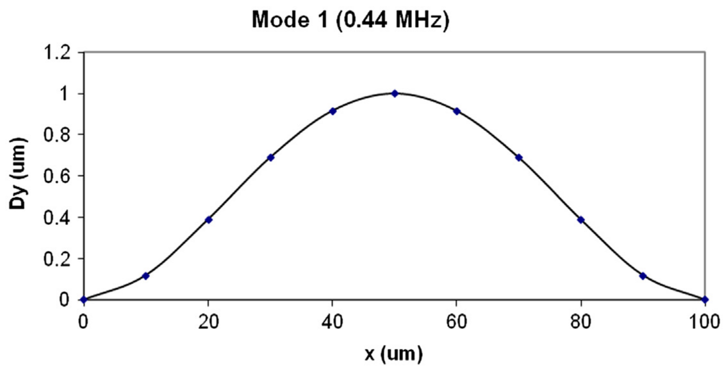

FIGURE 5.7

Mode 1 using 10 elements at 4.4285×10^5 Hz.

it will take a longer time to execute. Hence, it is advised to use the minimum number of elements which give the results of desired accuracy.

Other than the resonant frequencies, the mode shapes can also be obtained. Mode shapes can be considered to be the way in which the structure vibrates at a particular natural frequency. They correspond to the eigenvector of the finite element equation, just like the resonant frequencies correspond to the eigenvalues of the finite element equation. Mode shapes can be important in some applications, where the points of zero displacements, like the center of the beam in Figure 5.8, need to be identified for the installation of devices which should not undergo huge vibration.

The data for constructing the mode shape for each eigenvalue or natural frequency can be obtained from the displacement output for that natural frequency. Figures 5.7–5.9 show the mode shapes obtained by plotting the displacement components using 10 elements. The figures show how the clamped–clamped beam will vibrate at natural frequencies. Note that the output data usually consists of only the output at the nodes, and these are then used by the post-processor or any graph plotting applications to form a smooth curve. Most post-processors thus contain curve fitting functions to properly plot the curves using the data values.

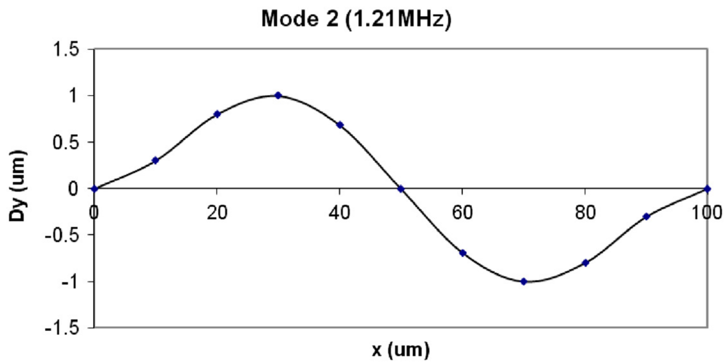

FIGURE 5.8

Mode 2 using 10 elements at 1.2284×10^6 Hz.

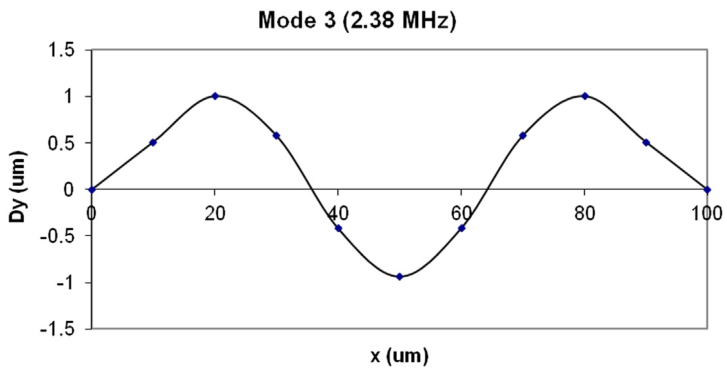

FIGURE 5.9

Mode 3 using 10 elements at 2.4276×10^6 Hz.

This simple case study points out some of the basic requirements needed in a finite element analysis. Like ABAQUS, most finite element software works on the same finite element principles. All that is needed is just to provide the necessary information for the software to use the necessary type of elements, and hence the shape functions; to build up the necessary element matrices; followed by the assembly of all the elements to form the global matrices; and finally, to solve the finite element equations. The next section in this case study will show a comparison with another popular FEM software package, ANSYS.

5.5.5 Comparison with ANSYS

In general, once one understands an FEM package well, it is generally much easier to learn than other packages. ANSYS is another popular FEM software package used widely by engineers. Although the user interface of ANSYS is different from that of ABAQUS, the FEM computation is essentially the same. Other differences between FEM software packages may include the use of different solvers. Because of these possible differences, the solutions can deviate from one another.

Most modelers using ANSYS run the simulation directly through its own software package via the graphical user interface (GUI). Nevertheless, the same information regarding the mesh (hence nodes and elements definition), material properties, beam section properties, analysis type, load, and boundary conditions are required by the software and they are keyed in through the GUI. Chapter 13 (Section 13.7) shows an example of setting up a simple cantilever beam model via the GUI in ANSYS. The procedures to set up this example model in ANSYS are similar to what is described in Chapter 13, except for the selection of the appropriate analysis type ("Modal" analysis is selected here, *cf.* Figures 13.12). Readers are referred to Chapter 13 to get an idea of how modeling can be done in ANSYS.

Figure 5.10 shows a screenshot of ANSYS with the deformation of the first mode plotted. The mode shape (eigenvector) of the first three modes obtained using ANSYS with 60 beam elements is the same as that obtained above using ABAQUS. From the inset window in Figure 5.10, the calculated natural frequencies for the first four modes are listed and they are found to be 4.4021×10^5 Hz, 1.2134×10^6 Hz, and 2.3786×10^6 Hz for modes 1, 2, and 3, respectively. While these values are close

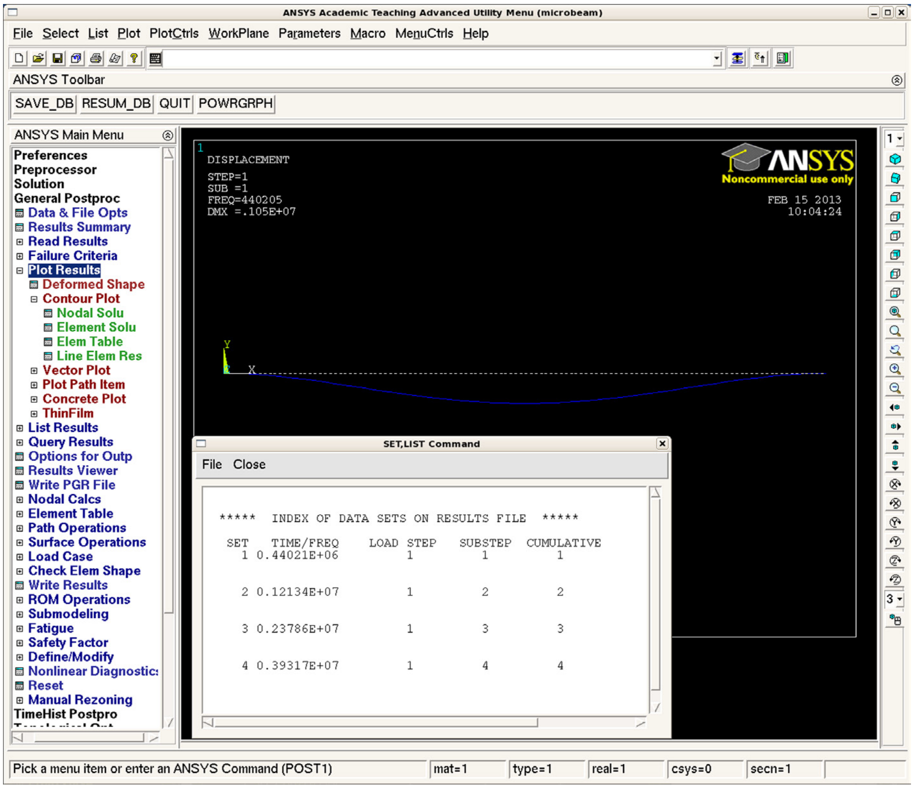

FIGURE 5.10

Screenshot of ANSYS showing the first eigenmode of the beam. The inset window shows the frequencies obtained using ANSYS.

to the analytical calculation in Table 5.3, some slight discrepancies in the frequencies can be observed. This can be explained by the different method chosen in ANSYS (amongst several options), to extract the eigenvalues for this problem. In this example, a Block Lanczos method is used to solve the eigenvalue problem, which can be as accurate as the subspace method, but is more efficient.

5.6 Review questions

1. How would you like to formulate a beam element that also carries axial forces?
2. Calculate the force vector for a simply supported beam subjected to a vertical force at the middle of the span, when only one beam element is used to model the beam, as shown in Figure 5.11.
3. Calculate the force vector for a simply supported beam subjected to a vertical force at the middle of the span, when two beam elements are used to model the beam, as shown in Figure 5.12.
4. Calculate the force vector for a simply supported beam subjected to a moment at the middle of the span, when one beam element is used to model the beam, as shown in Figure 5.13.
5. Figure 5.14 shows a uniform cantilever beam subjected to two vertical forces:
   a. Using one two-node beam element to model the entire beam, derive the external nodal force vector for the beam.
   b. Using two two-node elements of equal length, derive the external force vectors for these two elements.
   c. Discuss the accuracy of these two models.

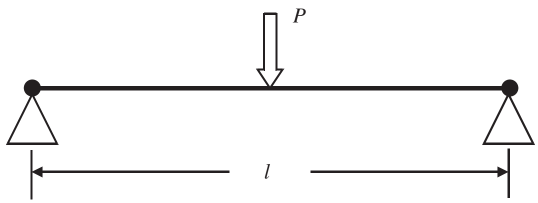

FIGURE 5.11

Simply supported beam modeled using one beam element.

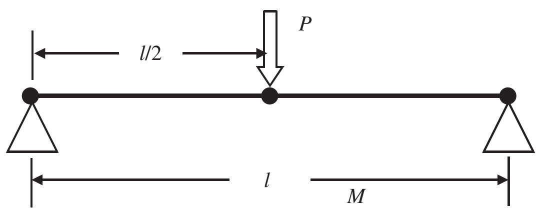

FIGURE 5.12

Simply supported beam modeled using two beam elements.

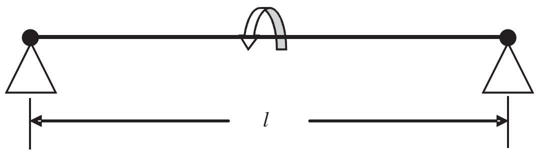

FIGURE 5.13

Simply supported beam modeled using one beam element.

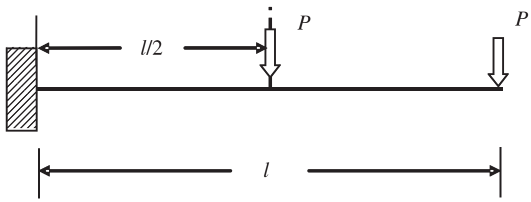

FIGURE 5.14

A cantilever beam subjected to two concentrated forces.

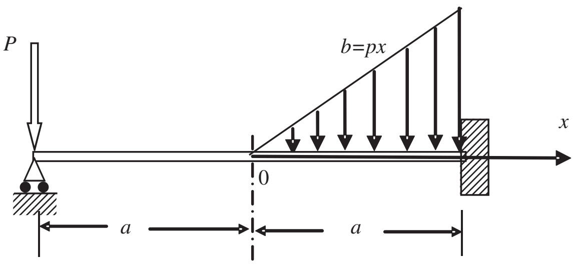

FIGURE 5.15

A beam subjected to concentrated and distributed forces.

6. Figure 5.15 shows a uniform beam subjected to a vertical concentrate force P and a distributed force $b = px$ on the right half span of the beam where p is a given constant.

- **a.** Using one two-node beam element to model the entire beam, derive the external nodal force vector for the beam.
- **b.** Using two two-node elements of equal length, give the formula to calculate the force vectors for these two elements (you are not required to carry out the integration for the right-hand side element, but full details of the integrand must be given).

7. For a cantilever beam subjected a vertical force at its free end, how many elements should be used to obtain the exact solution for the deflection of the beam?

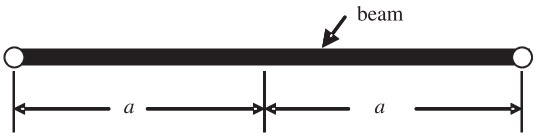

FIGURE 5.16

A beam with varying circular cross-section.

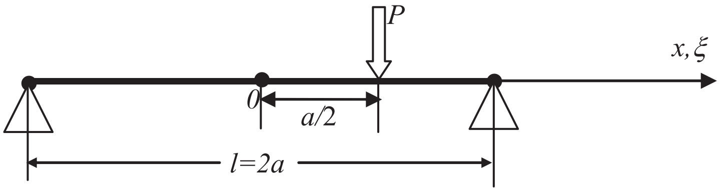

FIGURE 5.17

A simply supported beam.

8. Figure 5.16 shows a two-node beam of length $2a$. The beam is made of material of Young's modulus E and density ρ. The cross-section of the beam is circular, and the radius varies in the x-direction in a manner so that the area of the cross-section is a linear function of

$$A(x) = A_0 + \frac{A_1}{a}x.$$

a. Write down the expression for the element stiffness matrix.
b. Obtain k_{11} in terms of a, E, A_0, and A_1.
c. Write down the expression for the element mass matrix.
d. Obtain m_{11} in terms of a, ρ, A_0, and A_1.

9. Consider a beam with uniform rectangular cross-section, as shown in Figure 5.17. It is simply supported and subjected to a vertical load P at $x=a/2$. Using two elements for the entire beam, make use of the symmetry of the structure, and let $l=2$ m, $E=69$ GPa and $\upsilon=0.3$, $b=0.1$ m, $h=0.06$ m, and

$$I_z = \frac{1}{12}bh^3$$

a. Calculate the deflection v_c and rotation θ_c at the central point ($x=0$).
b. Calculate the rotations at two ends.