University of Nottingham
Department of Mechanical, Materials and Manufacturing Engineering

Computer Modelling Techniques

FE-02

SIMPLE 1D FINITE ELEMENTS

2.1 Introduction

Finite element methods can be introduced as a numerical procedure of analysing structural pin-jointed problems. In this section, the FE analysis of a simple one-dimensional (1D) pin-jointed element is first described and later extended to cover an assembly of 1D elements.

These pin-jointed structures consist of long thin elements linked together by frictionless pin-joints, which are assumed to transmit only axial forces to the elements. The elements do not bend. Elements that allow bending are referred to as “beam elements” which are more complex than pin-jointed elements.

Pin-jointed members are also referred to as “*trusses*”. Trusses are loaded only at the joints and the weight of the members may be neglected.

The analysis shown here is confined to only the x-direction (one-dimensional). The basic strategy of most FE formulations is **to** treat the nodal displacements (not the forces) as the unknown variables which can be determined by solving a system of linear algebraic equations.

2.2 A Simple Uniaxial 1D Pin-Jointed Element

The simplest type of element is a pin-jointed (bar) element subjected to an axial load. Figure 2.1 shows a straight uniaxial bar (or truss element) of length L_e and cross-sectional area A_e, with two points at either end (called "*nodes*"). The deformations (displacements) of nodes 1 and 2 are u_1 and u_2 respectively, with corresponding forces F_1 and F_2.

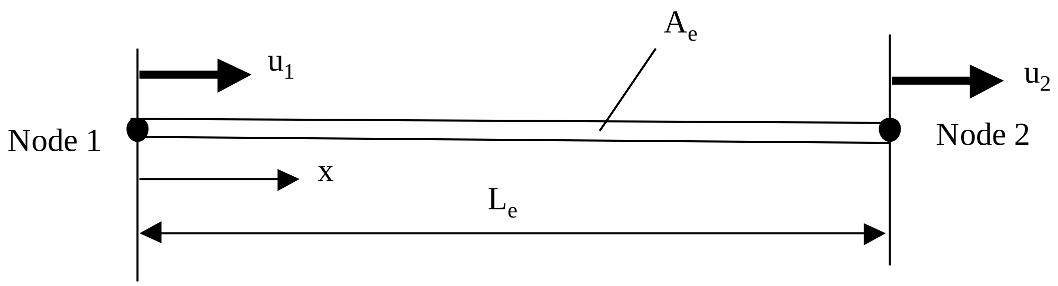

Figure 2.1: A one-dimensional bar element

The main strategy in formulating the FE equations is to derive an expression for the element '*stiffness*', i.e. treating the element as if it were a spring of stiffness k, as follows:

$$k\ u = F \tag{2.1}$$

where k is the stiffness, u is the displacement and F is the force.

Assuming that the material is linear elastic, the uniaxial stress-strain relationship is given by Young's modulus as follows:

$$\sigma = E\,\varepsilon \tag{2.2}$$

For a uniaxial bar, the strain is defined as the change in length divided by the original length, as follows:

$$\varepsilon = \frac{\Delta L}{Le} = \frac{u_1 - u_2}{Le} \tag{2.3}$$

Note that this definition of strain is simplistic, and only applies to a uniaxial long bar under tension or compression. In 2D and 3D continuum problems, a more sophisticated definition of strain must be used.

The stress in the bar is given by:

$$\sigma = \frac{F}{A_e} \tag{2.4}$$

Substituting the stress and strain in equation (2.2) results in:

$$\frac{F}{A_e E} = \frac{u_1 - u_2}{L_e} \tag{2.5}$$

Hence, a general force-displacement relationship can be obtained as follows:

$$F = \frac{A_e E}{L_e}(u_1 - u_2) \tag{2.6}$$

At nodes 1 and 2, the forces can therefore be expressed as two simultaneous equations, as follows:

$$\begin{aligned} \frac{A_e E}{L_e}(u_1 - u_2) &= F_1 \\ \frac{A_e E}{L_e}(u_2 - u_1) &= F_2 \end{aligned} \tag{2.7}$$

which can be expressed in matrix form as follows:

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix} \tag{2.8}$$

where

$$k_1 = \frac{E A_e}{L_e} \tag{2.9}$$

The matrix expression can be expressed in a more concise way as follows:

$$[k_e][u_e] = [F_e] \tag{2.10}$$

where $[k_e]$ is the "*element stiffness matrix*" (of size 2x2), $[u_e]$ and $[F_e]$ are the element displacement and force "vectors", respectively (each of size 2x1).

It is worth noting that the stiffness matrix is a symmetrical matrix. The symmetry is fortunate as it saves a considerable amount of computational time when using a numerical equation solving algorithm.

2.3 A More General Energy Derivation of Element Stiffness

The above derivation of the stiffness matrix is based on using equilibrium equations (equal and opposite forces F_1 and F_2). This works well for a 1D uniaxial element. However, in more sophisticated elements, such as 3D continuum elements or shell elements, using an equilibrium approach is very tedious and is not adaptable to other elements. For a more generalised approach that is applicable to all element geometries, the energy formulation is widely used to derive FE formulations.

The derivation of the FE formulation for any element of any geometry can be usually broken down into 7 main steps, as shown below.

Step 1: Define the element and the shape (interpolation) functions

The uniaxial bar can only carry loads in the axial direction, and is thus a one-dimensional element which cannot be used for problems with bending moments or shear loads. A uniaxial element is said to have one "*degree of freedom*" per node, i.e. only one independent variable (the uniaxial displacement).

The displacement, u, is always used as the independent variable in FE formulation, as it automatically satisfies the compatibility equations, i.e. the element faces move together with no gaps or overlaps. If forces are used as the independent variables, the elements will not be compatible.

For a 2-node element, we assume a linear displacement function of x as follows:

$$u = C_1 + C_2\,x \tag{2.11}$$

where C_1 and C_2 are constants. These constants can be expressed in terms of the nodal displacements (u_1 and u_2) by satisfying the element's boundary conditions, which are:

- At node 1 (where $x = 0$), $u = u_1$
- At node 2 (where $x = L_e$), $u = u_2$.

Therefore, the constants C_1 and C_2 can be expressed in terms of u_1 and u_2 as follows:

$$\begin{aligned} u_1 &= C_1 + 0 \quad \Rightarrow \quad C_1 = u_1 \\ u_2 &= C_1 + C_2\,L_e \quad \Rightarrow \quad C_2 = \frac{u_2 - u_1}{L_e} \end{aligned} \tag{2.12}$$

The displacement function can therefore be expressed in terms of the nodal displacements as follows:

$$u = u_1 + \left(\frac{u_2 - u_1}{L_e}\right)x \tag{2.13}$$

which can be rearranged as follows:

$$u = \left(1 - \frac{x}{L_e}\right)u_1 + \left(\frac{x}{L_e}\right)u_2 \tag{2.14}$$

This process is similar to curve-fitting where a straight line equation is obtained from the coordinates of two points at either end. The above equation (with u_1 and u_2 instead of C_1 and C_2) is physically more meaningful since it expresses the displacement of any point on the element as a function of the two displacements at the corner nodes.

The functions that multiply the nodal displacements u_1 and u_2 are called the "*shape functions*" or the "*interpolation functions*". For a 2-node element, the shape functions are linear. If more nodes are used per element, e.g. three nodes, the expression for the displacement function and the shape functions become quadratic.

Step 2: Satisfy the material law (constitutive equations)

The material law in this simple uniaxial problem is simply given by Young's modulus definition, as follows:

$$\sigma = E\,\varepsilon \tag{2.15}$$

Step 3: Derive the element stiffness matrix

Instead of using the equilibrium of the forces on the element, a more generalised energy derivation is used here. This energy approach is much more convenient than the force equilibrium approach, particularly when more complex 3D elements are used.

The principle of minimum total potential energy (T.P.E.) can be used to minimise the strain energy function with respect to the nodal displacements. The principle states that the TPE must be minimised with respect to the displacements. The TPE is expressed as follows:

$$T.P.E. = U - W \tag{2.16}$$

where U is the strain energy and W is the work done by the forces.

The strain energy, U, is given by:

$$U = \frac{1}{2}\,[\sigma]\,[\varepsilon] \times Volume \tag{2.17}$$

Therefore, for a uniaxial element of length L_e, the strain energy is:

$$U = \int_0^{L_e} \frac{1}{2}\,\sigma\,\varepsilon\,\left(A\,dx\right) \tag{2.18}$$

Using the more accurate definition of strain as a differential of the displacement function, the strain in this one-dimensional element is given by:

$$\varepsilon = \frac{du}{dx} \tag{2.19}$$

Note that since the strain is a function of only the x-coordinate, it is appropriate to use (du/dx) instead of the partial differentiation ($\partial u/\partial x$). By using equation (2.14), the strain can be expressed as follows:

$$\frac{du_x}{dx} = \left(\frac{-1}{L_e}\right)u_1 + \left(\frac{1}{L_e}\right)u_2 = \frac{u_2 - u_1}{L_e} \tag{2.20}$$

Substituting for stress in terms of strain from equation (2.15), the strain energy expression can be expressed as a function of the displacement as follows:

$$U = \int_0^{L_e} \frac{1}{2} (E\,\varepsilon)(\varepsilon) A\,dx$$
$$= \int_0^{L_e} \frac{1}{2} E\,(\varepsilon)^2 A\,dx$$
$$= \frac{E\,A}{2} \int_0^{L_e} \left(\frac{u_2 - u_1}{L_e} \right)^2 dx$$
$$= \frac{E\,A}{2\,L_e^{\;2}} (u_2 - u_1)^2 [x]_0^{L_e} \tag{2.21}$$
$$= \frac{E\,A}{2\,L_e} \left(u_2^{\;2} - 2\,u_2\,u_1 + u_1^{\;2} \right)$$

The work done by the forces F_1 and F_2 is simply expressed as the force multiplying the displacement at each node, as follows:

$$W = F_1\,u_1 + F_2\;u_2 \tag{2.22}$$

Therefore, the T.P.E. expression can be written in terms of the displacements as follows:

$$T.P.E = \frac{E\,A}{2\,L_e} \left(u_2^{\;2} - 2\,u_2\,u_1 + u_1^{\;2} \right) - \left(F_1\,u_1 + F_2\;u_2 \right) \tag{2.23}$$

Since there are two displacements in the TPE expression, the principle of minimum TPE requires minimisation with respect to both u_1 and u_2, which yields two equations. Minimising TPE with respect to u_1 gives:

$$\frac{\partial T.P.E.}{\partial u_1} = 0 = \frac{E\,A}{2\,L_e} \left(-\,2\,u_2 + 2\,u_1 \right) - F_1 \tag{2.24}$$

Similarly, minimising T.P.E. with respect to u_2 gives:

$$\frac{\partial T.P.E.}{\partial u_2} = 0 = \frac{E\,A}{2\,L_e} \left(2\,u_2 - 2\,u_1 \right) - F_2 \tag{2.25}$$

The above two equations can be rearranged as follows:

$$\begin{aligned} 0 &= \frac{E\,A}{L_e} \left(u_1 - u_2 \right) - F_1 \\ 0 &= \frac{E\,A}{L_e} \left(-\,u_1 + u_2 \right) - F_2 \end{aligned} \tag{2.26}$$

which can be combined in matrix form as follows:

$$\begin{bmatrix} k_1 & -\,k_1 \\ -\,k_1 & k_1 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix} \tag{2.27}$$

where $k_1 = EA/L_e$, which is identical to the expression derived using the force equilibrium in equation (2.8).

2.4 Element Assembly (More than one element)

All FE formulations start with the derivation of the stiffness matrix for a single element, and then combining the element with its neighbouring elements (element assembly). Two important relationships must be satisfied in the element assembly:

(i) The displacement of a particular node must be the same for every element connected to it.

(ii) The externally applied forces at the nodes on the surface must be balanced by the 'internal' forces on the elements at the nodes.

Step 4: Assemble the overall stiffness matrix

To demonstrate the assembly of elements in a FE mesh, consider a very simple mesh of two uniaxial bar (truss) elements (e_1 and e_2) joined together by a pin-jointed connection, as shown in Figure 2.2.

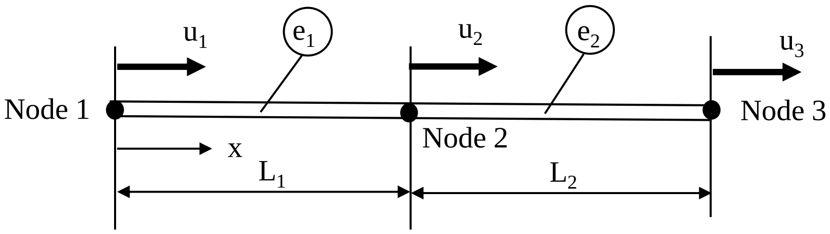

Figure 2.2: An element assembly of two uniaxial pin-jointed elements

There are 3 nodes in this mesh, with node 2 being common to both elements. For generality, we assume that the elements have different lengths, L_1 and L_2, and different stiffnesses, k_1 and k_2.

Taking each element in turn, as shown in Figure 2.3, the individual element stiffness matrices can be constructed and then assembled together.

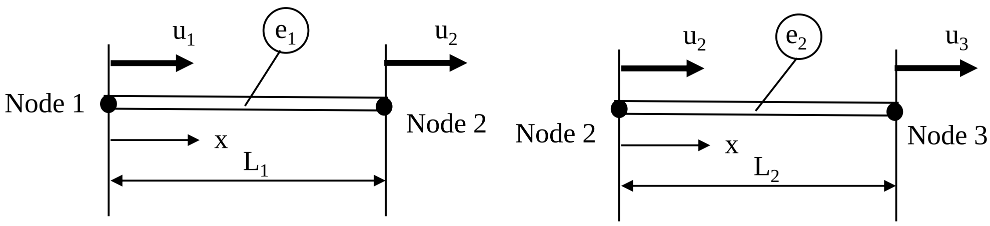

Figure 2.3: Individual elements in the assembly

For element e_1, the 'internal' force on node 1 is given by:

$$F_1 = k_1 \left(u_1 - u_2 \right) \tag{2.28}$$

Similarly, for element e_2, the 'internal' force on node 3 is given by:

$$F_3 = k_2 \left(u_3 - u_2 \right) \tag{2.29}$$

Since node 2 is shared by the two elements, the 'internal' F_2 can be written as two components:

$$\begin{aligned} \left(F_2\right)_{e_1} &= -F_1 = -k_1 \left(u_1 - u_2 \right) \\ \left(F_2\right)_{e2} &= -F_3 = -k_2 \left(u_3 - u_2 \right) \end{aligned} \tag{2.30}$$

Therefore, the total 'internal' force on node 2 is given by:

$$F_2 = \left(F_2\right)_{e_1} + \left(F_2\right)_{e_2} = -F_1 - F_3 = -k_1 \left(u_1 - u_2 \right) - k_2 \left(u_3 - u_2 \right) \tag{2.31}$$

Therefore, three simultaneous equations can be written for u_1, u_2 and u_3, as follows:

$$\begin{aligned} F_1 &= k_1 u_1 - k_1 u_2 \\ F_2 &= -k_1 u_1 + \left(k_1 + k_2\right) u_2 - k_2 u_3 \\ F_3 &= -k_2 u_2 + k_2 u_3 \end{aligned} \tag{2.32}$$

The above 3 equations can be assembled in matrix form as follows:

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \\ F_3 \end{bmatrix} \tag{2.33}$$

which can be written as a general equation for the stiffness of the assembly $[K]_{Assembly}$ (the whole FE mesh) as follows:

$$\left[K\right]_{Assembly} \left[u\right]_{Assembly} = \left[F\right]_{Assembly} \tag{2.34}$$

Again, note the symmetry of the assembly stiffness matrix.

Step 5: Apply the boundary conditions and external loads

The main objective of the FE formulation is to solve the simultaneous equations of the assembly to obtain the values of all nodal displacements (here u_1, u_2 and u_3). In this example, there are 3 simultaneous equations which cannot be solved until some nodal displacements or nodal forces are given.

Therefore, to obtain a 'unique' solution of the problem, some displacement constraints (called 'boundary conditions)' and some loading (force) conditions must be prescribed at some of the nodes. Without these conditions, the problem definition is incomplete. These conditions usually take one of the following forms:

(i) Prescribed displacement (called 'Boundary Condition'):
A zero or non-zero prescribed nodal displacement, or sliding against a rigid surface.

(ii) Prescribed load (force):
An applied force in a given direction or a prescribed pressure. Note that if a force is not prescribed at a given node, it is automatically assumed to have a prescribed nodal force of zero (i.e. a free surface).

The boundary conditions and loads can be incorporated into the simultaneous equations, which can then be solved to obtain a unique solution for the displacements at each node.

As an example, consider the problem shown in Figure 2.4, where the left node (node 1) is fixed to a rigid surface (i.e. $u_1 = 0$) , while a uniaxial force W is applied at the other end (i.e. $F_3 = W$ at node 3).

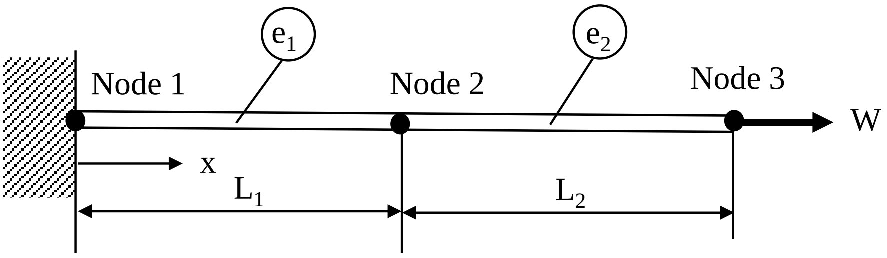

Figure 2.4: Element assembly example with boundary conditions and loads

Therefore, the prescribed values are:

(i) A prescribed displacement (i.e. boundary condition) of u_1=0.
Note that both u_2 and u_3 are unknown and will be calculated by solving the simultaneous equations.

(ii) A non-zero prescribed external force (load) of F_3=W.
Note that nodes 1 and 2 are automatically assigned a zero prescribed external force, i.e. F_1 =0 and F_2=0.

Implementing the above prescribed values in the assembly equation (2.33) gives:

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{bmatrix} 0 \\ u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ W \end{bmatrix} \qquad (2.35)$$

Since u_1 is given, the first equation is not required, and the assembly matrices shrink down to just two equations by eliminating the rows and columns that involve u_1, i.e. row 1 and column 1, as follows:

$$\begin{bmatrix} k_1+k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{bmatrix} u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} 0 \\ W \end{bmatrix} \qquad (2.36)$$

Step 6: Solve the simultaneous equations

In this simple problem of a 3x3 stiffness matrix, it is relatively straightforward to solve for the nodal displacements using algebraic manipulation. However, most FE practical applications contain thousands or hundreds of thousands of nodes, and can only be solved numerically.

Standard numerical equation solvers, such as the Gaussian elimination technique can be used to solve the equations to determine the unknown variables (here displacements) at each node.

To solve the simultaneous equations in equation (2.36), all geometry parameters (A, L_1 and L_2) must be known, together with the material properties (here only Young's modulus E). By solving the simultaneous equation by algebraic manipulation, the two unknown displacements can be obtained as follows:

$$\begin{aligned} u_2 &= \frac{W}{k_1} \\ u_3 &= \frac{W}{k_2} + \frac{W}{k_1} \end{aligned} \tag{2.37}$$

Step 7: Compute other variables

After solving the assembly equations, displacements at all the nodal points are determined. From the displacement values, the element strains can be obtained from the strain-displacement relationship in equation (2.19), and the element stresses from the material law in equation (2.15).

It worth emphasising again that, in most FE formulations, only the displacements are used as the independent variables. Other variables (such as forces, strains and stresses) are obtained from the computed displacements. For this reason, the computed FE displacements are usually (slightly) more accurate that the computed FE stresses.

2.5 Summary of Key Points

- In deriving the FE formulation, the first step is to define the order of variation (e.g. linear or quadratic) of displacement (not force) over each element.

- Displacements are the only ‘independent’ variables in FE formulations. All other variables (such as force, stress, strain, etc.) are derived from the displacements.

- A linear shape function can be used with 2-node pin-jointed elements. This means that the displacement is allowed to vary linearly per element, and the strain, which is a differential of the displacement, is therefore constant per element. For linear elastic analysis, the stress is also constant per element since it is linearly dependent on strain.

- For each element, an element stiffness expression can be derived as follows:
  $[k_e][u_e]=[F_e]$

- Two alternative approaches can be used to derive the element stiffness matrix; either a direct equilibrium (non-energy) approach, or a more general energy approach based on minimising the total potential energy (TPE). The energy approach is more versatile and can be easily adapted for other problems.

- In structural analysis problems, the individual element stiffness matrices are assembled together in the global stiffness matrix by combining the forces (and element stiffness) of the nodes which are shared between two or more elements.

- The element stiffness matrix and the overall stiffness matrix of the assembly are always symmetric.

- To obtain a unique solution of the simultaneous equations, boundary conditions and applied loads must be prescribed at some of the nodes.