7 Finite Element Method

Learning Summary

1. Recognise that FEA is a useful technique to aid the solution of many Structural Mechanics problems (knowledge);
2. Understand how 1D elements and the matrix method can be used to analyse uniaxial bars (application);
3. Apply the theory for 1D elements and the matrix method to an assembly of bars (application);
4. Understand the derivation of the global stiffness matrix of a truss element (knowledge);
5. Describe how 2D approximations can be used to simplify 3D problems (comprehension).

7.1 Introduction

The finite element (FE) method was first developed in the 1950s for the aerospace and nuclear power industries. At present, with the advancement of computing technology and the availability of a wide range of sophisticated commercial FE modelling software, the method is extensively used in the automotive industry and generally for structural and stress analyses in a variety of engineering parts and components as well as for steady and dynamic temperature distributions, fluid flow and manufacturing processes such as injection moulding and metal forming.

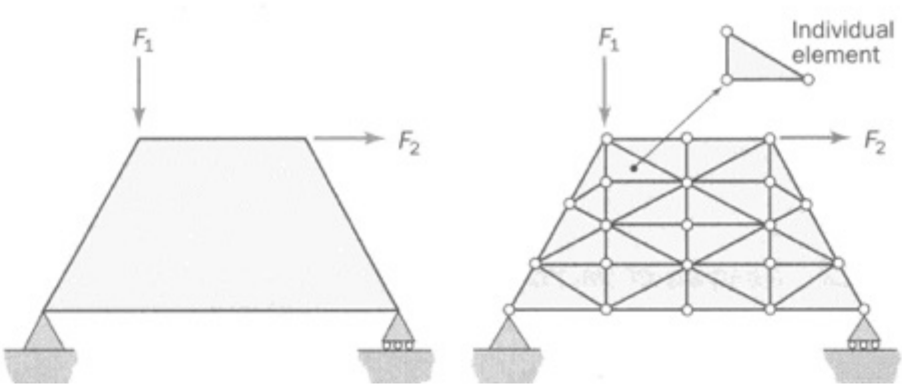


Figure 7.1: Finite Element Discretisation

Consider a plate, which is an elastic continuum, supported and loaded as shown in Figure 7.1. If the continuum was divided into a large number of parts or elements (triangular elements in this case), the stress distribution in the whole plate can be obtained by analysing each of the small elements in turn. To achieve this, it is necessary, for each element, to maintain equilibrium, deformation compatibility and stress-strain relationships. This forms the basis of the FE method. A few types of elements are shown below.

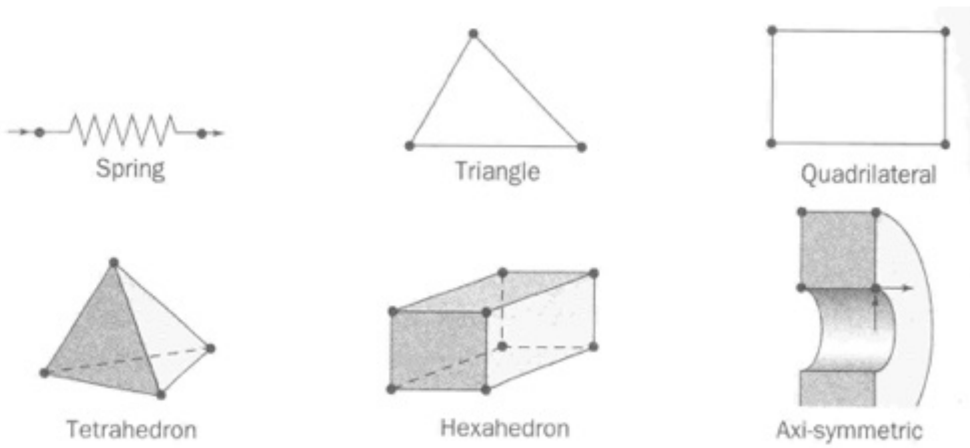
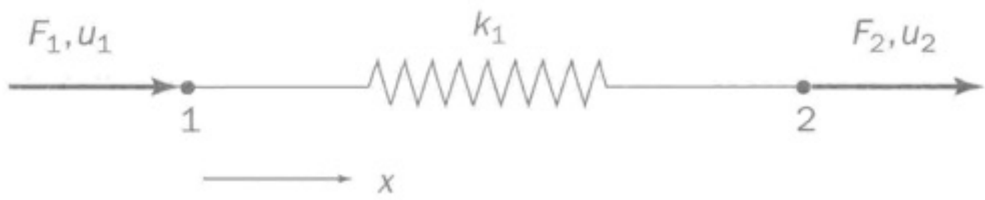


Figure 7.2: Types of finite elements

The accuracy of the FE solution depends on the number of elements. However, although the analysis of each element is relatively simple, the complete analysis for a large number of elements is extremely tedious, hence the necessity of computers.

7.2 Analysis of a spring element (one-dimensional)

The spring element shown can be used to model a uniform bar under uniaxial loading. The points of attachment to other parts or elements are called nodes (1 & 2). The nodal forces and displacements (F & u respectively) as well as the spring stiffness, k_1, are also shown. For a bar of length L, area A and elasticity modulus E, k_1 = AE/L.



For the spring (or bar) element, which is one-dimensional (1D), the forces and displacements are related by the following equations[1]:

$$F_1 = k_1(u_1 - u_2) = k_1u_1 - k_1u_2 \tag{7.1}$$

$$F_2 = k_1(u_2 - u_1) = -k_1u_1 + k_1u_2 \tag{7.2}$$

The displacements u_1 and u_2 are known as the degrees of freedom (DOFs).

If we express Equations (7.1) and (7.2) in matrix form we have:

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \tag{7.3}$$

Which may be written in shortand matrix form as:

$$\{F\} = [K^e]\{u\} \tag{7.4}$$

Where **[*K*^{e}] is the stiffness matrix for the spring element**. Note that it is symmetrical about the diagonal. In general, whether for an element or a complete structure, the stiffness matrix is always symmetrical.

7.3 Assembly of spring elements

Consider the arrangement of two spring elements as shown in Figure 7.3

[1] In these notes, vector quantities are denoted with braces { }, two dimensional arrays are contained in brackets [ ].

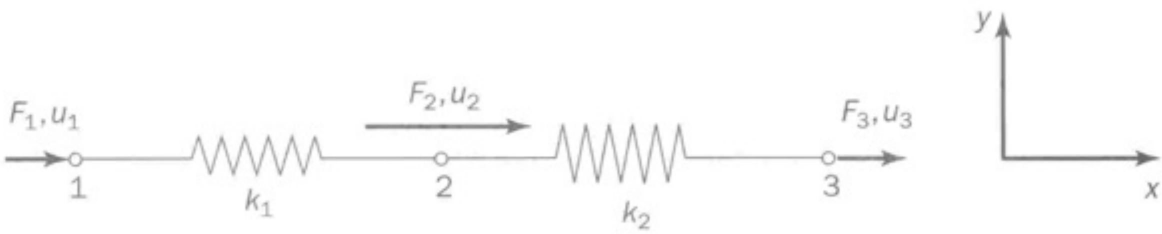


Figure 7.3: Assembly of two spring elements

Referring to equation (7.3), the force-displacement equation for each element is:

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \tag{7.5}$$

$$\begin{Bmatrix} F_2 \\ F_3 \end{Bmatrix} = \begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} \tag{7.6}$$

Expanding equations (7.5) and (7.6) to equivalent forms gives:

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} = \begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \tag{7.7}$$

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} = \begin{bmatrix} 0 & 0 & 0 \\ 0 & k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \tag{7.8}$$

The matrices can then be added to obtain the forces in the overall system:

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} = \begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \tag{7.9}$$

or in shorthand as:

$$\{F\} = [K]\{u\} \tag{7.10}$$

Here, **[*K*] is the global stiffness matrix for the structure** (again note that it is symmetrical about the diagonal), since the complete structure is made up of the two spring (or bar) elements. The global stiffness matrix has been assembled from the element stiffness matrices. In general, the global force-displacement equations for the whole assembly can be obtained by combining the stiffness matrix contributions of all the individual elements such that the [*K*] coefficients belonging to common nodes are added together.

- **If NNodes is the total number of nodes in the system and NDOF is the number of degrees of freedom per node then the global stiffness matrix is (NNodes x NDOF) by (NNodes x NDOF) in size.**
- **If sufficient boundary conditions are specified, the forces and displacements can be found. This approach to finding a solution is the basis of the FE method.**
- **For complex problems the accuracy of the solution is determined by the number of elements used (and the order of the elements, linear, quadratic etc.).**

7.4 Truss elements

While 1D spring elements are a good, simple introduction to demonstrate the matrix method and the concept of stiffness matrices, they have limited practical use. Truss elements are an extension of the spring element where each node has 2 degrees of freedom (DOFs) and they can be connected to form frame structures such as that shown in Figure 7.4

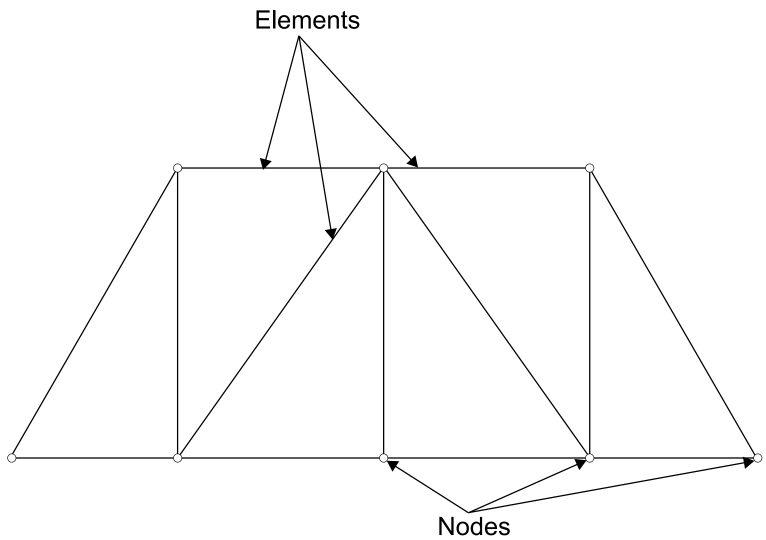


Figure 7.4: Truss element structure

The coordinate system for a truss element is shown in Figure 7.5

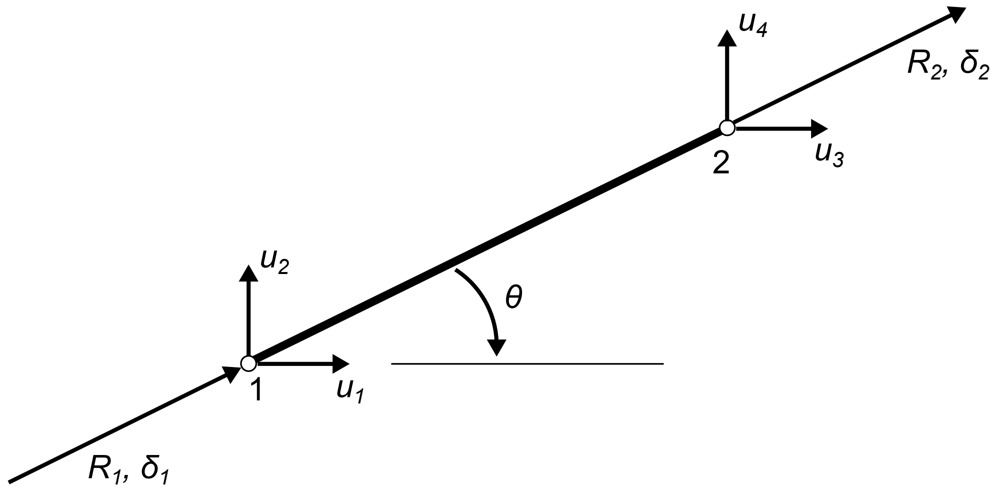


Figure 7.5: Truss element coordinate system

We need to establish a relationship between the local deflections, $\delta_1$and δ_2, and the global deflections u_1, u_2, u_3 and u_4.

$$\delta_1 = u_1 \cos\theta + u_2 \sin\theta \tag{7.11}$$

$$\delta_2 = u_3 \cos\theta + u_4 \sin\theta \tag{7.12}$$

Which can be expressed in matrix form (using $c = \cos\theta$ and $s = \sin\theta$) as:

$$\begin{Bmatrix} \delta_1 \\ \delta_2 \end{Bmatrix} = \begin{bmatrix} c & s & 0 & 0 \\ 0 & 0 & c & s \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} \tag{7.13}$$

Or in shorthand notation:

$$\{\delta\} = [T]\{u\} \tag{7.14}$$

Where [*T*] is the Transformation matrix.

We also need to relate the local forces, R_1 and R_2 to the global forces (in the same manner. The work done is the same in local and global coordinate systems:

$$W = \{\delta_1 \quad \delta_2\} \begin{Bmatrix} R_1 \\ R_2 \end{Bmatrix} = \{u_1 \quad u_2 \quad u_3 \quad u_4\} \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix} \tag{7.15}$$

Or in shorthand notation:

$$W = \{\delta\}^T\{R\} = \{u\}^T\{F\} \tag{7.16}$$

However substituting in for $\{\delta\}$ from Equation (7.30):

$$W = ([T]\{u\})^T\{R\} = \{u\}^T\{F\} \tag{7.17}$$

Or:

$$[T]^T\{u\}^T\{R\} = \{u\}^T\{F\} \tag{7.18}$$

Cancelling $\{u\}^T$ leaves:

$$[T]^T\{R\} = \{F\} \tag{7.19}$$

Recalling that the local stiffness equation for the truss element is:

$$\begin{Bmatrix} R_1 \\ R_2 \end{Bmatrix} = \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} \delta_1 \\ \delta_2 \end{Bmatrix} \tag{7.20}$$

Or in shorthand notation:

$$\{R\} = [K^e]\{\delta\} \tag{7.21}$$

Substituting Equation (7.21) into (7.19) for $\{R\}$ gives:

$$\{F\} = [T]^T[K^e]\{\delta\} \tag{7.22}$$

Substituting in Equation (7.14) for $\{\delta\}$ gives:

$$\{F\} = [T]^T[K^e][T]\{u\} \tag{7.23}$$

If we consider $[T]^T[K^e][T]$:

$$[T]^T[K^e][T] = \begin{bmatrix} c & 0 \\ s & 0 \\ 0 & c \\ 0 & s \end{bmatrix} \begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{bmatrix} c & s & 0 & 0 \\ 0 & 0 & c & s \end{bmatrix} \tag{7.24}$$

Multiplying through gives:

$$[T]^T[K^e][T] = k\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \tag{7.25}$$

Recalling that $k = AE/L$ gives the element stiffness matrix for a truss element in the global coordinate system as follows:

$$[K^e]_g = \left(\frac{AE}{L}\right)\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \tag{7.26}$$

It should be noted that this stiffness matrix is also symmetrical about the diagonal and is 4 x 4 in size as each element has 4 unknown displacements. Again $c = \cos\theta$ and $s = \sin\theta$.

7.5 Principle of virtual work (PVW)

Consider a 2D body in equilibrium under the action of a set of vector forces, $\vec{F}_i$, giving rise to a set of vector displacements and therefore to an internal stress distribution, $\underline{\sigma}(x, y)$, and strain distribution, $\underline{\varepsilon}(x, y)$ as shown in Figure 7.6

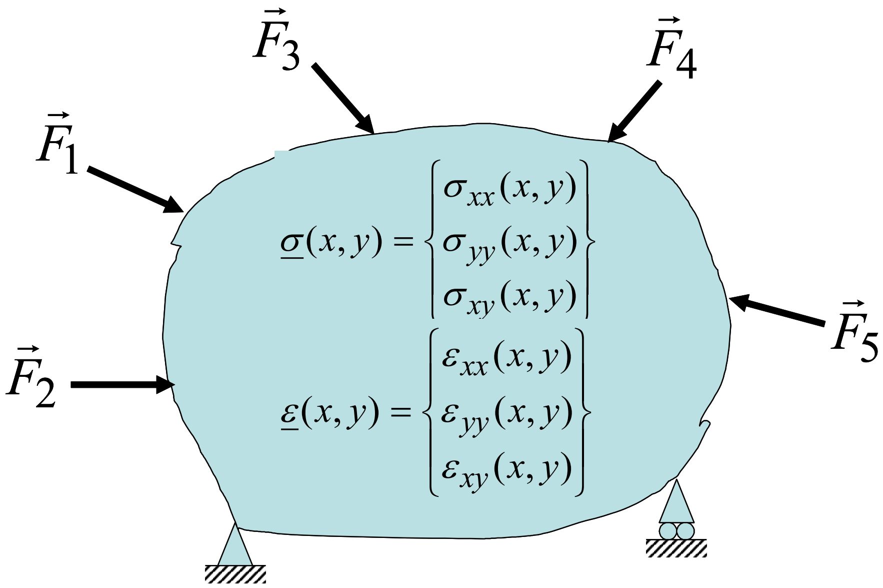


Figure 7.6: 2D body in equilibrium

The principle of virtual work (PVW) says that the work done by a set of forces in equilibrium moving through a set of small, compatible displacements is zero.

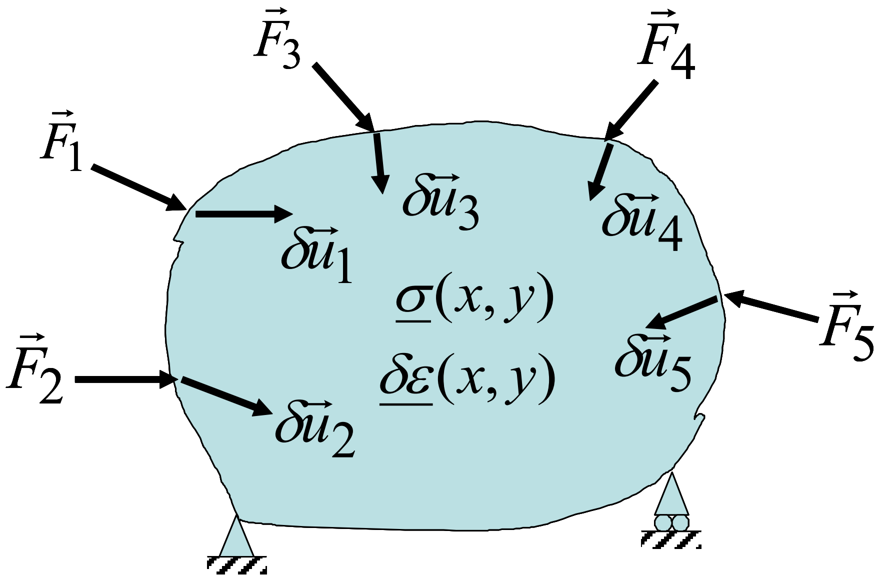


Figure 7.7: 2D body showing small displacements

For deformable bodies, the total virtual work consists of the sum of external virtual work and internal virtual work:

$$\delta W = \delta W_{int} + \delta W_{ext} \tag{7.27}$$

where:

$$\delta W_{int} = -\int_V \left(\delta\varepsilon_{xx}\sigma_{xx} + \delta\varepsilon_{yy}\sigma_{yy} + \delta\varepsilon_{xy}\sigma_{xy}\right)dV \tag{7.28}$$

$$\delta W_{ext} = \delta\vec{u}_1\vec{F}_1 + \delta\vec{u}_2\vec{F}_2 + \delta\vec{u}_3\vec{F}_3 + \delta\vec{u}_4\vec{F}_4 + \delta\vec{u}_5\vec{F}_5 \tag{7.29}$$

The principle of virtual work is fundamental to structural finite element (FE) analysis, as shown below. It permits the development of expressions for the stiffness matrices of a range of different structural element types. In turn this permits computational modelling of complex geometries to give approximate solutions for displacement, stress and strain distributions.

7.6 1D example of the finite element method (FEM)

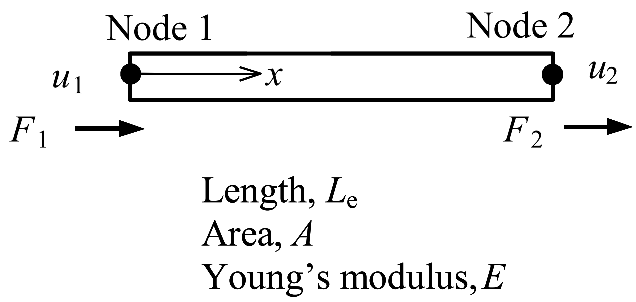


Figure 7.8: 1D element

The 1D structure is divided up into elements, connected together at nodes as shown in Figure 7.8. The displacement pattern can be expressed in terms of a linear polynomial

$$u = \alpha_1 + \alpha_2 x \tag{7.30}$$

where α_1 and α_2 are constants, determined from the nodal displacements and element geometry.

At node 1, x = 0, so

$$u = u_1 = \alpha_1 \tag{7.31}$$

and at node 2, $x = L_e$, so

$$u = u_2 = \alpha_1 + \alpha_2 L_e \tag{7.32}$$

Substituting equation (7.31) into (7.32) and rearranging:

$$\alpha_2 = \frac{(u_2 - u_1)}{L_e} \tag{7.33}$$

Hence the resultant variation in displacement over the element is, from equation (7.30):

$$u = u_1 + \frac{(u_2 - u_1)}{L_e} x \tag{7.34}$$

or:

$$u = \left(1 - \frac{x}{L_e}\right) u_1 + \frac{x}{L_e} u_2 \tag{7.35}$$

The displacement over the element can be given in the following terms:

$$u(x) = u_1 N_1(x) + u_2 N_2(x) \tag{7.36}$$

or in matrix form:

$$u(x) = [N_1(x) \quad N_2(x)] \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \tag{7.37}$$

$$u(x) = [N]\{u\} \tag{7.38}$$

where u_1, u_2 are the nodal degrees of freedom (dofs) and $N_1(x)$, $N_2(x)$ are called the **shape functions** of the element.

From equation (7.35) it can be seen that in this case:

$$N_1(x) = \left(1 - \frac{x}{L_e}\right) \tag{7.39}$$

$$N_2(x) = \frac{x}{L_e} \tag{7.40}$$

and that the displacement is linear as shown in Figure 7.9.

.

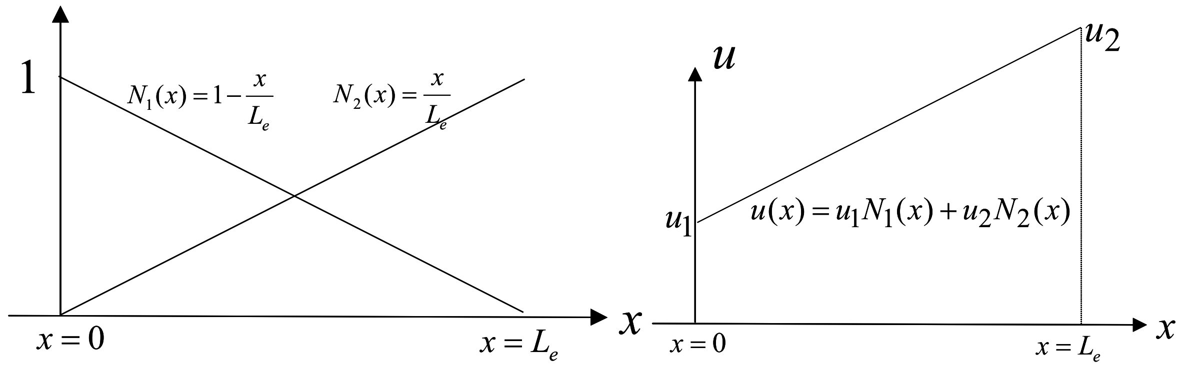


Figure 7.9: Element shape functions and displacement over the element

The only strain we are concerned with is the strain in the x-direction:

$$\varepsilon_x = \frac{du}{dx} = \frac{d}{dx}[N]\{u\} = \frac{d}{dx}\begin{bmatrix} 1 - \frac{x}{L_e} & \frac{x}{L_e} \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \tag{7.41}$$

Only the shape functions vary with *x* and so equation (7.41) becomes:

$$\varepsilon_x = \begin{bmatrix} -\frac{1}{L_e} & \frac{1}{L_e} \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \tag{7.42}$$

which can be written as:

$$\varepsilon_x = [B]\{u\} \tag{7.43}$$

where [*B*] is the strain-displacement matrix, or beta matrix and gives the strain at any point in the element due to the nodal displacements.

The stress-strain (constitutive) relation for a uniaxial bar is:

$$\sigma = E\varepsilon \tag{7.44}$$

and substituting equation (7.43) into (7.44) gives:

$$\sigma = [E][B]\{u\} = E\begin{bmatrix} -\frac{1}{L_e} & \frac{1}{L_e} \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \tag{7.45}$$

Internal virtual work:

$$\delta W_{\text{int}} = -\int_0^{L_e} \delta \underline{\varepsilon}^T \underline{\sigma} A \, dx$$
$$= -A \int_0^{L_e} \{\delta \underline{u}\}^T \{\underline{B}\}^T E \{\underline{B}\} \{\underline{u}\} \, dx$$
$$= -EA \{\delta \underline{u}\}^T \int_0^{L_e} \{\underline{B}\}^T \{\underline{B}\} \, dx \{\underline{u}\}$$

External virtual work:

$$\delta W_{ext} = \delta u_1 P_1 + \delta u_2 P_2 = \{\delta \underline{u}\}^T \{\underline{P}\}$$

Principle of virtual work (PVW)

$$\delta W_{ext} + \delta W_{\text{int}} = 0$$

$$EA \{\delta \underline{u}\}^T \int_0^{L_e} \{\underline{B}\}^T \{\underline{B}\} \, dx \{\underline{u}\} = \{\delta \underline{u}\}^T \{\underline{P}\}$$
$$\Rightarrow EA \int_0^{L_e} \{\underline{B}\}^T \{\underline{B}\} \, dx \{\underline{u}\} = \{\underline{P}\}$$
$$\Rightarrow \{K\} \{\underline{u}\} = \{\underline{P}\}$$

This is the stiffness equation for the element of a 1D stress system:

$$\{K\} = EA\int_0^{L_e} \{\underline{B}\}^T \{\underline{B}\} dx$$

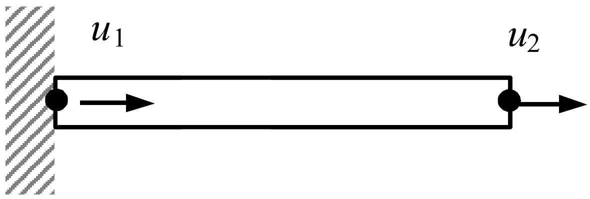
$$= EA\int_0^{L_e} \begin{Bmatrix} -\frac{1}{L_e} \\ \frac{1}{L_e} \end{Bmatrix} \begin{Bmatrix} -\frac{1}{L_e} & \frac{1}{L_e} \end{Bmatrix} dx$$

$$= \frac{EA}{L_e} \begin{Bmatrix} 1 & -1 \\ -1 & 1 \end{Bmatrix}$$

$$\Rightarrow \frac{EA}{L_e} \begin{Bmatrix} 1 & -1 \\ -1 & 1 \end{Bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix}$$

7.6.1 Solution of stiffness equation

To solve the stiffness equation boundary conditions must be applied, otherwise elements can move as rigid bodies and we cannot solve the equations for unique values of displacement.



Suppose $u_1 = 0$, then $u_2 = \frac{P_2 L_e}{EA}$ which is the exact solution for this problem. This is because the assumed linear variation of displacement is the exact variation in this case.

Once the primary variables have been solved for, by solving the stiffness matrix, then back-substitution into the strain-displacement and stress-strain relations yields the strains and stresses in each element and consequently in the complete FE model for a multi-element analysis.

Strain:

$$\Rightarrow \varepsilon = \left\{ \begin{array}{cc} -\frac{1}{L_e} & \frac{1}{L_e} \end{array} \right\} \left\{ \begin{array}{c} u_1 \\ u_2 \end{array} \right\} = \left\{ \underline{B} \right\} \left\{ \begin{array}{c} u_1 \\ u_2 \end{array} \right\} = B_2 u_2 = \frac{1}{L_e} \frac{P_2 L_e}{EA} = \frac{P_2}{EA}$$

Stress:

$$\sigma = E \left\{ \underline{B} \right\} \left\{ \begin{array}{c} u_1 \\ u_2 \end{array} \right\} = E\varepsilon = E \frac{P_2}{EA} = \frac{P_2}{A}$$

Reactions:

$$\frac{EA}{L_e} \left\{ \begin{array}{cc} 1 & -1 \\ -1 & 1 \end{array} \right\} \left\{ \begin{array}{c} u_1 \\ u_2 \end{array} \right\} = \left\{ \begin{array}{c} P_1 \\ P_2 \end{array} \right\} = \left\{ \begin{array}{c} R \\ P \end{array} \right\}$$

$$u_1 = 0$$

$$\Rightarrow -\frac{EA}{L_e} u_2 = R$$

$$\Rightarrow R = -\frac{EA}{L_e} \frac{PL_e}{EA} = -P$$

7.7 Two-dimensional finite element analysis

The methods of one-dimensional analysis apply equally to two-dimensional analysis. The main change is an increase in the number of nodes and of nodal variables due to the extra dimension. Of course all problems are three-dimensional but it can save significant time and effort if a two-dimensional assumption can be made. Two such assumptions are the ***plane stress*** and ***plane strain*** assumptions.

7.7.1 Plane stress assumption

If we have a thin plate which is only loaded in the in-plane directions, i.e. the only forces are F_x and F_y, then because the normal stress σ_z must be zero on the front and back faces, i.e. $\sigma_z = 0$, and because the plate is thin, then we can assume that $\sigma_z ==0$ throughout the thickness. Thus, the only non-zero components of stress are σ_x, σ_y and τ_{xy} and we can determine all of the strain components, i.e. ε_x, ε_y, ε_z and γ_{xy}, from these stress components, using Hooke's law for elastic behaviour, for example.

$$\varepsilon_x = \frac{1}{E}\left(\sigma_x - \nu\sigma_y\right)$$

$$\varepsilon_y = \frac{1}{E}\left(\sigma_y - \nu\sigma_x\right)$$

$$\varepsilon_z = \frac{-\nu}{E}\left(\sigma_x + \sigma_y\right)$$

$$\gamma_{xy} = \frac{\tau_{xy}}{G}$$

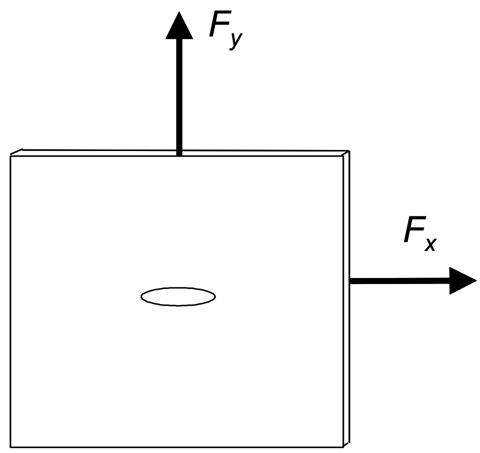


Figure 7.10

Note the plane stress assumption also applies to the free surfaces of components.

7.7.2 Plane strain assumption

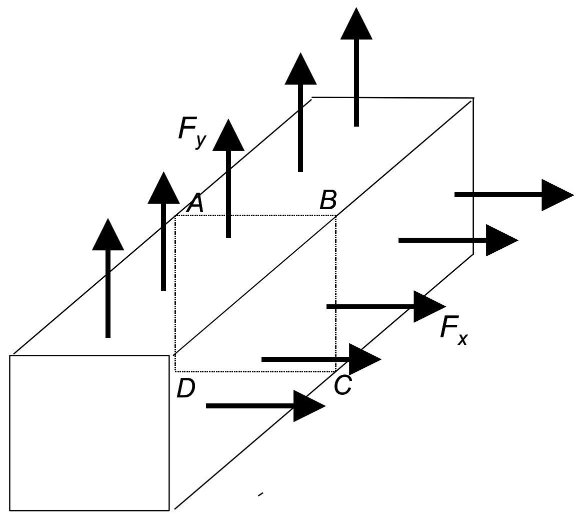


Figure 7.11

If we have a very thick plate or a very long member of a regular cross-section in the *z*-direction and if it is again only loaded in the in-plane directions, i.e. the only forces are F_x and F_y, then we can assume that a plane ABCD, which is remote from the ends, experiences negligible strain in the *z*-direction, i.e. ε_z ==0. Thus, for elastic behaviour, using Hooke's law:

$$\varepsilon_z = 0 \Rightarrow \sigma_z = \nu(\sigma_x + \sigma_y)$$

so the *z*-direction stress can be determined from the *x*- and *y*-direction normal stresses, so that this solution also allows a two-dimensional approach.

1.1 Constant strain triangle

A simple 2D element is the Constant Strain Triangle (CST) element shown in Figure 7.12. This is a 3-noded triangular element that allows linear variation of displacements as follows:

$$u_x(x,y) = C_1 + C_2 x + C_3 y$$
$$u_y(x,y) = C_4 + C_5 x + C_6 y$$

where u_x, u_y are the *x*- and *y*-direction displacements.

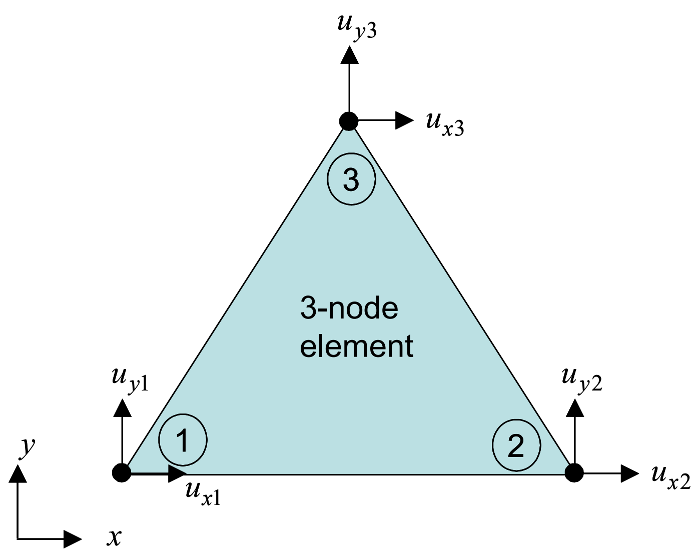


Figure 7.12: The Constant Strain Triangle (CST) Element

It can be shown that the constants C_1 to C_6 are related geometrically to the nodal coordinates (x_i,y_i) and the nodal values of $u_x(x,y)$ and $u_y(x,y)$ so that

$$\begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \\ u_{x3} \\ u_{y3} \end{bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x_1 & y_1 \\ 1 & x_2 & y_2 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x_2 & y_2 \\ 1 & x_3 & y_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x_3 & y_3 \end{bmatrix} \begin{bmatrix} C_1 \\ C_2 \\ C_3 \\ C_4 \\ C_5 \\ C_6 \end{bmatrix}$$

$$\Rightarrow \left[\underline{u}_e\right] = \left[A\right]\left[C\right] \qquad \text{(A)}$$

The strains are as follows:

$$\varepsilon_x = \frac{\partial u_x}{\partial x} = C_2 \quad \varepsilon_y = \frac{\partial u_y}{\partial y} = C_6$$

$$\gamma_{xy} = \frac{\partial u_x}{\partial y} + \frac{\partial u_y}{\partial x} = C_3 + C_5$$

Clearly these strains are all constant values; hence the name constant strain triangle (CST).

Thus

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 1 & 0 & 1 & 0 \end{bmatrix} \begin{bmatrix} C_1 \\ C_2 \\ C_3 \\ C_4 \\ C_5 \\ C_6 \end{bmatrix}$$

i.e. $[\underline{\varepsilon}] = [X][C]$

But $[C] = [A]^{-1}[u_e]$ from Eq (A)

$$\Rightarrow [\underline{\varepsilon}] = [X][A]^{-1}[\underline{u}_e] = [\underline{B}][\underline{u}_e] \qquad \text{(B)}$$

where $[\underline{B}]$ is again the strain-displacement matrix, or the beta matrix. In this case, $[\underline{B}]$ is a constant matrix, the entries of which are made up of product terms involving 0, 1 and the nodal coordinates (x_i,y_i).

For plane stress, the stress-strain relationship for elastic behaviour is:

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} \frac{1}{E} & \frac{-\nu}{E} & 0 \\ \frac{-\nu}{E} & \frac{1}{E} & 0 \\ 0 & 0 & \frac{2(1+\nu)}{E} \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix}$$

A similar relationship exists for the plane strain case.

The inverse of this gives

$$[\underline{\sigma}] = [\underline{D}][\underline{\varepsilon}]$$

$$\Rightarrow [\underline{\sigma}] = [\underline{D}][\underline{B}][\underline{u}_e] \qquad \text{(C)}$$

where $[D]$ is referred to as the constitutive matrix, the material matrix or the stress-strain matrix.

As for the one-dimensional elements, the element stiffness behaviour is

$$[\underline{P}_e] = [K_e][\underline{u}_e]$$

where

$$[K_e] = \int_{vol_e} [\underline{B}]^T [\underline{D}][\underline{B}] dV \qquad \text{(D)}$$

i.e. an integral over the volume, *vol*e, of each element, using the strain-displacement and stress-strain (constitutive) matrices. This integration is normally carried out using a numerical integration scheme called Gaussian quadrature, due to the complexity of the integrand for more complex elements.

Element assembly, paying attention to

(a) compatibility of nodal displacements, and
(b) nodal force equilibrium

leads to the global stiffness equation for a complete FE model, as follows:

$$[\underline{P}_g] = [K_g][\underline{u}_g]$$

This can be solved numerically by inverting $\left[K_g\right]$ and pre-multiplying by $\left[\underline{P}_g\right]$ thus

$$\left[\underline{u}_g\right] = \left[K_g\right]^{-1}\left[\underline{P}_g\right] \qquad \text{(E)}$$

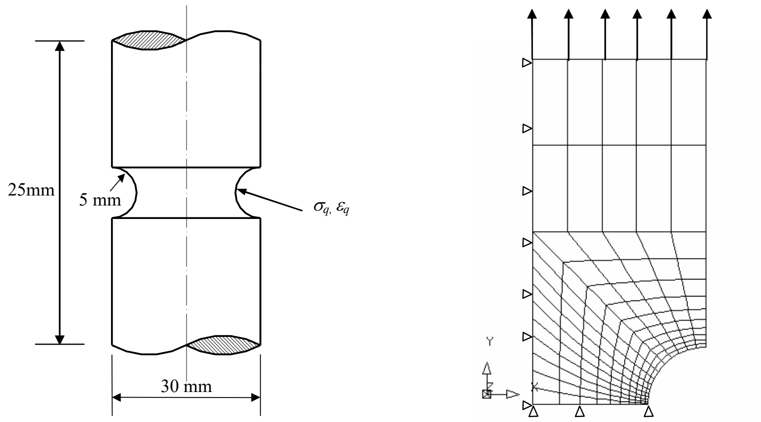
The external force vector $\left[\underline{P}_g\right]$ is normally specified in terms of point loads or distributed loads. The stiffness matrix $\left[K_g\right]$ is obtained from the individual element stiffness matrices which are derived from Eq (D) which uses the element geometry, material properties and the shape functions.

The global displacement vector $\left[\underline{u}_g\right]$ is thus obtained from Eq (E) and this gives the individual element displacement vectors $\left[\underline{u}_e\right]$. The element strains $\left[\underline{\varepsilon}\right]$ and stresses $\left[\underline{\sigma}\right]$ are then obtained from Eqs (B) and (C), i.e.

$$\left[\underline{\varepsilon}\right] = \left[\underline{B}\right]\left[\underline{u}_e\right]$$

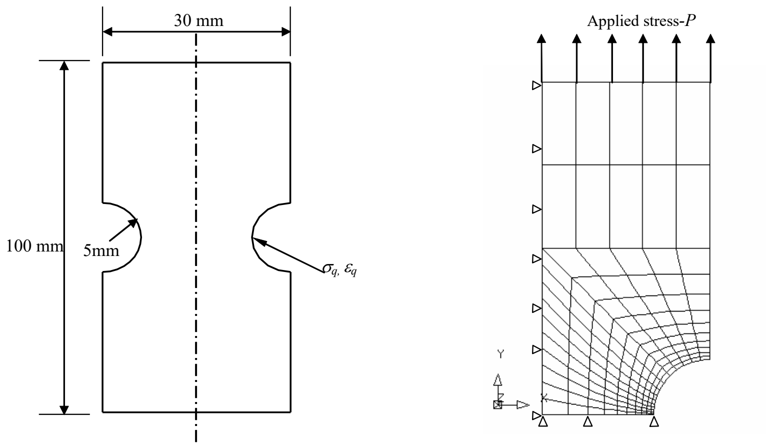
$$\left[\underline{\sigma}\right] = \left[\underline{D}\right]\left[\underline{B}\right]\left[\underline{u}_e\right]$$

7.8 Some typical FE models of notched members



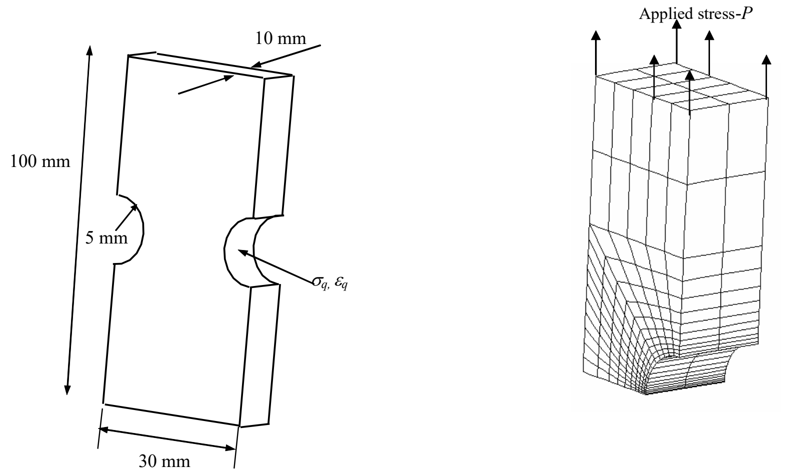
(a) Geometry (b) FE model

Figure 7.13: Geometry of round bar with a circumferential notch and mesh of axisymmetric finite elements



(a) Geometry (b) FE model

Figure 7.14: Geometry and mesh of 2D finite elements



(a) Geometry (b) FE model

Figure 7.15: Geometry and mesh of 3D finite element