FINITE ELEMENT PRINCIPLES

B.V.Smith

School of Electronic and Electrical Engineering,
University of Birmingham

1.0 INTRODUCTION

The Finite-Element Method (FEM) is essentially a numerical technique for the solution of partial differential equations applied to bounded continua. The engineering application has its origins in structures and much of the terminology reflects this, but the method is far more general and may be applied equally to many other physical and engineering problems. The main aim of this paper is to present the mathematical basis of FEM.

At least five distinct steps can be identified in the method:-

i) **Discretisation of the domain** - the bounded domain or region under study is divided up into many smaller regions; the elements. These elements usually have a regular geometrical form. A discrete number of points within the element or on its perimeter are specified; these are the nodes at which the physical parameter of interest is discretised. It is this complete set of nodal or discretised values which are the unknowns.

ii) **Choice of shape functions** - the physical parameter within an element is approximated throughout the element by interpolation between the nodal values. The functions of position which define the interpolation are called shape functions.

iii) **Derivation of element equations** - the derivation of a set of matrix equations which govern the behaviour of the individual elements is a key process in the FEM. In elastic continua, where the discretised parameter is usually the displacements, these equations are derived from considerations of equilibrium. However, in more general applications, such as fields, the equations are derived using either the method of weighted residuals or variational methods.

iv) **Assembly of elements** - this is the process whereby equilibrium is considered over the whole domain, ie. globally. Equilibrium at a particular node is obtained by summing contributions from all elements, with that node common. Combining the element matrices into a global array to give this overall equilibrium is called assembling.

v) **Solution of assembled equations** - the final step is the solution of the assembled equations to evaluate the set of unknown nodal values. Use of the interpolation functions together with these nodal values then enables the unknown physical parameter to be determined throughout the domain.

FINITE ELEMENT PRINCIPLES

In this paper the above five steps are applied to a simple one-dimensional problem (an elastic bar) to illustrate the essential features of FEM. Initially for comparison purposes the element equations for this case are derived by three different techniques; virtual work, weighted residuals (Galerkin) and minimisation of a variational function.

The element equations for a piezoelectric element are also introduced and applied to a simple axi-symmetric example (a thin-walled piezoceramic cylinder) to illustrate this important type of problem.

2.0 ELEMENT EQUATIONS

In the following discussions an elastic continuum will be considered (with the displacements as the unknowns) because of the direct relevance of this case to SONAR transducer structures. The displacement will be given by the matrix **u** (bold scripts will be used throughout to represent matrices), where this is a column matrix or vector with generally three components. In FEM the approximation arises when **u** is represented in terms of interpolation functions (for convenience in this text an equals sign will be used throughout even though an approximation is implied in many instances):-

$$\mathbf{u} = \mathbf{N}\mathbf{a} \; , \qquad (2.1)$$

where **a** are the set of nodal displacement vectors and **N** is the shape function matrix whose components will generally be a function of the co-ordinate system.

Generally undamped element equations of the following form will be obtained:-

$$\mathbf{F} = \mathbf{K}\mathbf{a} + \mathbf{M}\ddot{\mathbf{a}} \; , \qquad (2.2)$$

where **F** is the nodal (element boundary) applied force, **K** is the stiffness matrix, **M** is the mass matrix and $\ddot{\mathbf{a}}$ are the nodal accelerations. This effectively represents a force balance condition. In the case of sinusoidally alternating forces the steady-state (single frequency) equations become from (2.2):-

$$\mathbf{F} = \mathbf{K}\mathbf{a} - \omega^2\mathbf{M}\mathbf{a} \; , \qquad (2.3)$$

where ω is the frequency in radians per second. If frictional damping were included there would be an additional imaginary term on the right-hand side of (2.3).

The element equations will now be derived by the three techniques referred to in Section 1.0.

FINITE ELEMENT PRINCIPLES

2.1 Virtual Work

Consider there are small virtual displacements at the element nodes, given by $\delta\mathbf{a}$. The virtual work done on the element in the presence of the applied nodal forces $\mathbf{F}$ will therefore become the scalar product $\delta\mathbf{a}^t\mathbf{F}$, where the superscript t represents the transpose of the matrix. In the absence of frictional damping this work will be absorbed throughout the element in the re-distribution of the element strain, $\mathbf{e}$, as a change in the potential energy and in the re-distribution of the element accelerations, $\ddot{\mathbf{u}}$, as a change in the kinetic energy, according to the following:-

$$\text{strain energy} = \int_V \delta\mathbf{e}^t\boldsymbol{\sigma}dV$$

$$\text{inertia energy} = \int_V \delta\mathbf{u}^t\rho\,\ddot{\mathbf{u}}dV ,$$

where V is the element volume, ρ is the density of the material of the element and $\boldsymbol{\sigma}$ is the element stress. Now let the element strain be related to the displacement by the differential matrix transformation, $\mathbf{D}$, ie. $\mathbf{e} = \mathbf{Du}$. For example, in the one-dimensional case $\mathbf{D}$ becomes the scalar differentiator d/dx. Substituting for $\mathbf{u}$ from (2.1) gives:-

$$\mathbf{e} = \mathbf{DNa} = \mathbf{Ba} , \quad \delta\mathbf{e} = \mathbf{B}\,\delta\mathbf{a} , \quad \delta\mathbf{e}^t = (\mathbf{B}\,\delta\mathbf{a})^t = \delta\mathbf{a}^t\mathbf{B}^t$$

and $$\delta\mathbf{u}^t = (\mathbf{N}\,\delta\mathbf{a})^t = \delta\mathbf{a}^t\mathbf{N}^t.$$

Also let the element stress be related to the strain by $\boldsymbol{\sigma} = \mathbf{Ee}$, where $\mathbf{E}$ is the material stiffness matrix; thus the stress can be written as $\boldsymbol{\sigma} = \mathbf{EBa}$. Finally, for steady-state conditions $\ddot{\mathbf{u}} = -\omega^2\mathbf{Na}$.

Equating the strain and inertia energies to the virtual work and substituting the above relationships the following equation results:-

$$\delta\mathbf{a}^t\mathbf{F} = \delta\mathbf{a}^t\left(\int_V \mathbf{B}^t\mathbf{EB}dV\right)\mathbf{a} - \omega^2\,\delta\mathbf{a}^t\left(\int_V \mathbf{N}^t\rho\,\mathbf{N}dV\right)\mathbf{a}$$

From this it follows that:-

$$\mathbf{F} = \mathbf{Ka} - \omega^2\mathbf{Ma} , \qquad (2.4)$$

where, the element stiffness matrix, $\mathbf{K} = \int_V \mathbf{B}^t\mathbf{EB}dV$ (2.4a) and the element mass matrix, $\mathbf{M} = \int_V \mathbf{N}^t \rho \mathbf{N} dV$ (2.4b)

Consider applying the above equations to a simple one-dimensional element, namely an elastic-bar element of length, h, area, A, density ρ and material stiffness given by Young's modulus, E. Let the two ends of the bar be the nodes, designated by subscripts 1 and 2, so that the nodal force vector, **F**, has two components, F_1 and F_2. Also let the nodal displacement vector, **a** have the two components, a_1 and a_2. Assuming linear interpolation between the nodes and with the bar aligned along the x-axis, then the displacement vector, **u**, has a single component which becomes:-

$$u = (1-x/h)a_1 + (x/h)a_2 = N_1 a_1 + N_2 a_2 ,$$

where the shape function, **N**, is a row matrix, given by $[N_1, N_2]$.

For this one-dimensional case **D** = d/dx and therefore **B** becomes:-

$$\mathbf{B} = \mathbf{DN} = (d/dx)\,[N_1, N_2] = [-1/h\ ,\ 1/h] = (1/h)\,[-1, 1]$$

Substituting the matrices, **B** and **N**, and their transposes into equations (2.4a) and (2.4b), then (2.4) has the following two component equations (note, the volume integrals reduce to line integrals with the area, A, taken outside the integrals):-

$$F_1 = (AE/h)(a_1 - a_2) - \omega^2 \rho A \int_0^h [(1-x/h)^2 a_1 + (1-x/h)(x/h) a_2]dx$$

$$F_2 = (AE/h)(-a_1 + a_2) - \omega^2 \rho A \int_0^h [(1-x/h)(x/h) a_1 + (x/h)^2 a_2]dx$$

Solving the integrals, which are straightforward, the element equations become for this case:-

$$\begin{bmatrix} F_1 \\ F_2 \end{bmatrix} = (EA/h) \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \end{bmatrix} - m\omega^2 \begin{bmatrix} 1/3 & 1/6 \\ 1/6 & 1/3 \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \end{bmatrix} \quad (2.5)$$

where $m = \rho Ah$, the element mass.

2.2 <u>Weighted Residuals</u>

This method of determining the element equations is less

FINITE ELEMENT PRINCIPLES

specific than the virtual work method whose roots are firmly in structures. It is applicable to problems in which a physical parameter, say **u**, although in this case it is not necessarily displacement, obeys a partial differential equation set:-

L[u]=0 , where **L**[ ] is a set of linear partial differential operators.

If the parameter, **u**, is now approximated by **Na**, then the operation:-

L[Na] will no longer be zero, but will have a residual value. The idea of the weighted residual method is then to scalar multiply the residual by weighting functions, **W**, such that the integral of the product taken over the domain, H, is zero, ie.:-

$$\int_H \mathbf{W}^t \mathbf{L}[\mathbf{Na}]dH = 0 \quad ,$$

where the integral is over the domain. Now the integral of the sum of many parts is equal to the sum of the integrals of the parts, thus if the above weighted residual form is applied to an individual element then the assembly of these elements by summation will give the above integral.

There are several choices of weighting function, but the one most commonly encountered in FEM is the Galerkin form in which the weighting functions are the element shape functions. Thus for a single differential equation, L[u], the Galerkin form, applied to an element of volume V, becomes:-

$$\int_V N_i L[\mathbf{Na}]dV = 0 \quad ,$$

where i takes on values equal in number to the number of shape functions, which in turn equals the number of components of **a**.

Consider applying this method to the elastic-bar element for which the governing differential equation is of the Helmholtz form:-

$$L[u] = E\,(d^2/dx^2)u + \rho\omega^2 u = 0 \quad ,$$

where $\sqrt{E/\rho}$ is the sound velocity of compressional waves in the bar. Therefore applying the weighted residual technique to L[u] gives a pair of equations:-

FINITE ELEMENT PRINCIPLES

$$\int_V N_1 L[\mathbf{Na}]dv = 0 \quad \text{and} \quad \int_V N_2 L[\mathbf{Na}]dv = 0 \text{ , these become:-}$$

$$EA \int_0^h N_1[(d^2/dx^2)\mathbf{Na}]dx + A\rho\omega^2 \int_0^h N_1\mathbf{Na}dx = 0$$

$$EA \int_0^h N_2[(d^2/dx^2)\mathbf{Na}]dx + A\rho\omega^2 \int_0^h N_2\mathbf{Na}dx = 0$$

Integrating the first terms in these integral equations by parts produces two boundary terms:-

$$EAN_1(d/dx)\mathbf{Na}\Big|_0^h = F_1$$

$$EAN_2(d/dx)\mathbf{Na}\Big|_0^h = F_2$$

where F_1 and F_2 are the respective forces applied at the two nodes. Substituting for these boundary terms the pair of integral equations become:-

$$EA \int_0^h [(d/dx)N_1][(d/dx)\mathbf{Na}]dx - A\rho\omega^2 \int_0^h N_1\mathbf{Na}dx = F_1$$

$$EA \int_0^h [(d/dx)N_2][(d/dx)\mathbf{Na}]dx - A\rho\omega^2 \int_0^h N_2\mathbf{Na}dx = F_2$$

Now, $\mathbf{Na} = N_1a_1 + N_2a_2$, $N_1 = 1-x/h$ and $N_2 = x/h$. Substituting

these expressions into the above equations and evaluating the straightforward integrals that result, the pair of element equations, (2.5), are again obtained.

2.3 Variational Method

The main idea behind this method is to obtain a functional, say Φ, for the problem, which is minimised such that variations of the functional, $\delta\Phi$, with respect to the physical parameters, say a, are equated to zero, ie.:-

FINITE ELEMENT PRINCIPLES

$$\delta\Phi = (\partial\Phi/\partial a)\,\delta a = 0\,.$$

Since δ**a** may take any value, then $(\partial\Phi/\partial$ **a**$) = 0$. Application of this latter result for each of the components of **a** produces the required set of equations.

Consider the case of the element of an elastic-bar and form the following integral around the wave equation for a small variation, δu:-

$$I = A\int_0^h [E\,(d^2/dx^2)u + \rho\omega^2 u]\,\delta u\,dx$$

This integral can be equated to zero since the wave equation equals zero. Now the following applies:-

$$d/dx(\,du/dx\,\delta u) = (du/dx)[d/dx(\delta u)] + (d^2u/dx^2)\,\delta u$$

The last of these differential terms appears in I and so can be replaced by the first two terms. This yields a perfect differential which can be integrated to give:-

$$I = AE\,\delta u\,du/dx\Big|_0^h + A\int_0^h \{\rho\,\omega^2 u\delta u - E du/dx[(d/dx)\,\delta u]\}dx = 0$$

The boundary term can be written:-

$$AE\,\delta u\,du/dx\Big|_0^h = F_1\,\delta u_1 - F_2\,\delta u_2 = \delta W_1 - \delta W_2 = \delta(W_1 - W_2),$$

where W_1 and W_2 are the works done at the two nodes. Now making use of the following two results:-

$u\,\delta u = \delta(u^2)/2$ and $du/dx[(d/dx)\,\delta u] = \delta[(du/dx)^2]/2$, the integral I becomes:-

$$I = \delta\{\,(W_1 - W_2) + (A/2)\int_0^h [\rho\,\omega^2 u^2 - E\,(du/dx)^2]dx\} = \delta\{\,\Phi\,\} = 0,$$

where Φ is the functional. Incidently the two terms within the integral in the above equation may be recognised as the kinetic and potential energies within the element and hence the expression within the curley braces is essentially a statement of the conservation of energy. Now the nodal works may be written as:- $W_1 = F_1 a_1$ and $W_2 = F_2 a_2$.

FINITE ELEMENT PRINCIPLES

Therefore minimising the functional Φ :-

$$\partial\Phi/\partial a_1 = \partial/\partial a_1 (W_1 - W_2) +$$

$$+ \partial/\partial a_1 \{(A/2) \int_0^h [\rho \omega^2 u^2 - E\,(du/dx)^2] dx\} = 0$$

and there will be a second similar equation for $\partial\Phi/\partial a_2 = 0$. Replacing u by $(N_1 a_1 + N_2 a_2)$ in the above, carrying out the differentiations and the integrations, again generates the element equations (2.5).

3.0 FEM EXAMPLE APPLIED TO ELASTIC BAR

Consider the case of an elastic bar, of overall length, H, which is subjected to an alternating force at one end, is rigidly clamped at the other and is laterally free. Assume that the bar is aligned with the x-axis such that the force is applied at x=0 and the rigid clamping occurs at x=H. Let the amplitude of the alternating force be F_0 and its frequency be such that the length of the bar is a quarter of an acoustic wavelength, λ_a, ie. $H = \lambda_a/4$. The aim is to calculate the displacements along the bar under these circumstances.

This is a problem which has a simple analytic solution and hence the theoretical results can be compared with the results obtained by FEM. The bar is in fact resonant at its fundamental length mode and the amplitude of the displacement, as a function of x, is given by $u_1\cos(x\pi/2H)$, where u_1 is the amplitude of the displacement at x=0.

The first step is to divide the bar into a number of elements. The more elements used the more accurate the results, but at the cost of more calculation. As a compromise the bar is divided into four elements, which enables the essential features of FEM to be demonstrated without the calculations becoming excessive and confusing. The nodes and elements are numbered as is shown in Fig.1.

The element equations applicable to this problem have been derived above, set (2.5). They can be rewritten:-

$$\begin{bmatrix} F_i \\ F_j \end{bmatrix} = \begin{bmatrix} E_1 & -E_2 \\ -E_2 & E_1 \end{bmatrix} \begin{bmatrix} a_i \\ a_j \end{bmatrix} = \begin{bmatrix} \mathbf{E_{12}} \end{bmatrix} \begin{bmatrix} a_i \\ a_j \end{bmatrix}$$

FINITE ELEMENT PRINCIPLES

where i and j refer to the node numbers, $E_1 = [(EA/h) - (m\omega^2/3)]$ and $E_2 = [(EA/h) + (m\omega^2/6)]$. Therefore the equations for the four elements become:-

element (1) $$\begin{bmatrix} F_0 \\ F_2 \end{bmatrix} = \begin{bmatrix} E_{12} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \end{bmatrix}$$, where $F_1 = F_0$

element (2) $$\begin{bmatrix} F_2 \\ F_3 \end{bmatrix} = \begin{bmatrix} E_{12} \end{bmatrix} \begin{bmatrix} a_2 \\ a_3 \end{bmatrix}$$

element (3) $$\begin{bmatrix} F_3 \\ F_4 \end{bmatrix} = \begin{bmatrix} E_{12} \end{bmatrix} \begin{bmatrix} a_3 \\ a_4 \end{bmatrix}$$

element (4) $$\begin{bmatrix} F_4 \\ F_5 \end{bmatrix} = \begin{bmatrix} E_{12} \end{bmatrix} \begin{bmatrix} a_4 \\ 0 \end{bmatrix}$$, where $a_5 = 0$

F_1 and a_5 equal the prescribed boundary conditions at x=0 and x=H respectively. Also the element length, h, equals H/4.

When the above element equations are assembled (equivalent to taking equilibrium over the length of the bar) the boundary force associated with an internal node, eg. F_2 of element (1), will be equal and opposite to the boundary force at that same node from an adjacent element, eg. F_2 of element (2). Under these circumstances the components of force associated with internal nodes in the global matrix will be zero. Also, there is a basic requirement that there is continuity of displacement at each node. The assembled global matrix equation becomes:-

$$\begin{bmatrix} F_0 \\ 0 \\ 0 \\ 0 \\ F_5 \end{bmatrix} = \begin{bmatrix} E_1 & -E_2 & 0 & 0 & 0 \\ -E_2 & 2E_1 & -E_2 & 0 & 0 \\ 0 & -E_2 & 2E_1 & -E_2 & 0 \\ 0 & 0 & -E_2 & 2E_1 & -E_2 \\ 0 & 0 & 0 & -E_2 & E_1 \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \\ 0 \end{bmatrix}$$

where F_5 is the unknown force at node 5, which represents the reaction force at the rigid boundary and the nodal displacements, a are the other unknowns. The dotted lines have been drawn in to show the element origin of the various stiffness components. Two points to note are that the matrix is symmetrical and banded; a general result for this FEM displacement formulation. This reduces both storage requirements and speed of solution.

The solution of these equations whilst tedious is quite straightforward and leads to the following displacements:-

$$a_2 = F_0 \frac{E_2[2E_1 - (E_2^2/2E_1)]}{E_1\{[2E_1 - (E_2^2/E_1)][2E_1 - (E_2^2/2E_1] - E_2^2\}}$$

$$a_1 = (F_0 + E_2a_2)/E_1$$

$$a_3 = E_2a_2/[2E_1 - (E_2^2/2E_1)]$$

$$a_4 = E_2a_3/2E_1 \qquad (3.1)$$

The solution of these equations can now be carried out for both the static and the dynamic cases.

3.1 Static case

The results can be obtained for the static case by simply setting the frequency to zero, ie. $\omega = 0$; then $E_1 = E_2 = EA/h$. Substituting these into the set of equations (3.1) above, the following result:-

$a_1=4F_0h/EA$, $a_2=3F_0h/EA$, $a_3=2F_0h/EA$ and $a_4=F_0h/EA$.

These are the expected static displacements for an applied force F_0, eg. $a_1/4h$ is the static strain at x=0, F_0/A is the applied stress and E is the stiffness.

3.2 Dynamic case

For the dynamic case, $\omega = 2\pi c / \lambda_a$, where $c^2=E/\rho$, the bar sound-velocity, and $\lambda_a= 4H$. Therefore:-

$$\omega^2 = \pi^2E/4H^2\rho .$$

Substituting into E_1 and E_2 this expression for ω^2 and the expression for the element mass, $m = \rho Ah$, then the following expressions result:-

$$E_1 = 0.9486(EA/h) \quad \text{and} \quad E_2 = 1.0257(EA/h) .$$

Using these in equations (3.1) the following expressions for the

FINITE ELEMENT PRINCIPLES

displacements are obtained:-

a_1 = 256.72(F_0h/EA), a_2/a_1 = 0.921, a_3/a_1 = 0.704

and a_4/a_1 = 0.380.

These three ratios may be compared with the expected values of; $\cos(\pi/8) = 0.924$, $\cos(\pi/4) = 0.707$ and $\cos(3\pi/8) = 0.383$. The agreement with the nodal values obtained by FEM is quite good, but because of the linear interpolation, assumed between the nodes, the values of displacement at other positions would not be in such good agreement. This is illustrated in Fig.2 where the two solutions are compared graphically.

It is worth noting that the values of displacement obtained in the dynamic case are of the order of eighty times bigger than the static values. This occurs because of the resonance condition with no damping.

4.0 PIEZOELECTRIC ELEMENT EQUATIONS

The essential difference between the piezoelectric and non-piezoelectric case is the occurrence of a dielectric stiffness matrix and mixed stiffness matrices, which produce modifications to the conventional stiffness matrix. In addition the electric potential is discretised aswell as the displacement.

To derive the element equations the virtual work approach of Section 2.1 can be followed. Consider the following pair of piezoelectric equations which relate the elastic stress, $\boldsymbol{\sigma}$, the strain, **e**, the electric field strength, $\boldsymbol{\mathcal{E}}$, and the charge density, **q** :-

$$\boldsymbol{\sigma} = \mathbf{E}\mathbf{e} - \mathbf{e}_p \boldsymbol{\mathcal{E}} \qquad (4.1a)$$

$$\mathbf{q} = \mathbf{e}_p^t \mathbf{e} + \boldsymbol{\epsilon} \boldsymbol{\mathcal{E}} \qquad (4.1b)$$

where **E** is the material elastic stiffness (as previously), $\mathbf{e}_p$ and its transpose are piezoelectric parameters, and $\boldsymbol{\epsilon}$ is the electric permittivity.

Consider the virtual work associated with small nodal virtual-variations in elastic displacement, $\delta\mathbf{a}$, and in electrical potential $\delta\boldsymbol{\phi}$, together with applied nodal forces, **F**, and applied nodal charge **Q**:-

$$\delta\mathbf{a}^t\mathbf{F} = \int_V \delta\mathbf{e}^t\boldsymbol{\sigma}dV + \int_V \delta\mathbf{u}^t\rho\,\ddot{\mathbf{u}}dV \qquad \text{(elastic)} \qquad (4.2a)$$

FINITE ELEMENT PRINCIPLES

$$\delta\phi^t Q = \int_V \delta\mathcal{E}^t q\,dV \qquad \text{(electric)} \qquad (4.2b)$$

Replace the stress in (4.2a) by (4.1a), replace the charge density in (4.2b) by (4.1b) and make use of the following relationships:-

$$e = Ba, \quad \delta e^t = \delta a^t B^t, \quad \mathcal{E} = -D_e\phi \text{ and } \delta\mathcal{E}^t = -\delta\phi^t D_e{}^t,$$

where D_e includes a differential matrix operating on the shape function for the potential. Then following the procedure used in Section 2.1 equations (4.2) become:-

$$F = K_{uu}a + K_{u\phi}\phi - \omega^2 Ma \qquad (4.3a)$$

$$-Q = K_{\phi u}a + K_{\phi\phi}\phi, \qquad (4.3b)$$

where the "stiffness" and mass matrices are:-

$$K_{uu} = \int_V B^t E B\,dV \qquad \text{(elastic stiffness)}$$

$$K_{u\phi} = \int_V B^t e_p D_e\,dV \qquad \text{(piezoelectric "stiffness")}$$

$$K_{\phi u} = \int_V D_e{}^t e_p{}^t B\,dV \qquad \text{(inverse piezoelectric "stiffness")}$$

$$K_{\phi\phi} = -\int_V D_e{}^t \in D_e\,dV \qquad \text{(electric "stiffness")}$$

$$M = \int_V N^t \rho N\,dV \qquad \text{(mass)}$$

Substituting the potential from (4.3b) into (4.3a) this latter equation is condensed:-

$$F^1 = K^1 a - \omega^2 Ma \quad , \qquad (4.4)$$

where the modified force and stiffness matrices are:-

FINITE ELEMENT PRINCIPLES

$$F^1 = F + K_{u\phi}(K_{\phi\phi})^{-1}Q$$

and

$$K^1 = K_{uu} - [K_{u\phi}(K_{\phi\phi})^{-1}K_{\phi u}]$$

Equation (4.4) is of identical form to the element equations (2.4), but with mixed matrices. So for the piezoelectric element, equation (4.4) can be assembled and solved in the same way as the elastic element equations, but with the boundary conditions for charge aswell as force applied. Once the displacements have been evaluated the potential distribution can be obtained from (4.3b).

5.0 AXI-SYMMETRIC APPLICATIONS

There are many important types of SONAR transducer which exhibit cylindrical symmetry and so this application will now be discussed.

Consider a cross-section through an axi-symmetric structure, such as the cylinder shown in Fig.3. Let the cylinder have a length, h, inner radius, R and let it be described by cylindrical co-ordinates, r,z,θ. The small three-noded triangle represents an arbitrary finite element. Let the nodal displacements in the z and r directions respectively, be a and b,

where $a = [a_1,a_2,a_3]^t$, $b = [b_1,b_2,b_3]^t$

and the subscripts are the node numbers.

Now in FEM the general z-displacement, u, within the element, is represented:-

$$u = [f_1,f_2,f_3] \begin{bmatrix} a_1 \\ a_2 \\ a_3 \end{bmatrix}$$

where f_1,f_2,f_3 are suitable interpolation functions of position r,z. For the three-noded triangle these are usually simple linear functions. Similarly for the r-displacement, v,:-

$$v = [f_1,f_2,f_3] \begin{bmatrix} b_1 \\ b_2 \\ b_3 \end{bmatrix}$$

Therefore u and v can be written:-

$$\begin{bmatrix} u \\ v \end{bmatrix} = [If_1,If_2,If_3] \begin{bmatrix} a \\ b \end{bmatrix} = N_u \begin{bmatrix} a \\ b \end{bmatrix}$$

FINITE ELEMENT PRINCIPLES

where **I** is the unit matrix and $\mathbf{N}_u$ is the shape function for displacement.

For this axi-symmetric case the strains, e, become:-

$$\mathbf{e} = \begin{bmatrix} \partial/\partial z & 0 \\ 0 & \partial/\partial r \\ 0 & 1/r \\ \partial/\partial r & \partial/\partial z \end{bmatrix} \begin{bmatrix} u \\ v \end{bmatrix} = \mathbf{D} \begin{bmatrix} u \\ v \end{bmatrix} = [e_z, e_r, e_\theta, e_{rz}]^t$$

Now the matrix **B**, which appears in the element stiffness-matrix, is given by $\mathbf{B} = \mathbf{DN}_u$. Since f_1, f_2, f_3 are simple linear functions of r,z, then **B** will have constant coefficients, except where these are associated with the 1/r coefficient in **D**, ie. those associated with the circumferential strain, e_θ. (It is worth noting that for a thin planar element, where there are only two strain components e_r and e_z, all of the coefficients of **B** are independent of position co-ordinates, ie. a constant strain element.)

Once **B** has been determined then the stiffness matrix is given by:-

$$\mathbf{K}_{uu} = \int_V \mathbf{B}^t\mathbf{EB}dV = \int_V \mathbf{B}^t\mathbf{EB}r\,dr\,dz\,d\theta \ ,$$

but for cylindrical symmetry **B** is independent of θ and hence:-

$$\mathbf{K}_{uu} = 2\pi \int_A \mathbf{B}^t\mathbf{EB}r\,dr\,dz \ ,$$

where for an isotropic material in cylindrical co-ordinates the material stiffness matrix, **E**, is:-

$$\frac{E}{(1+\mu)(1-2\mu)} \begin{bmatrix} 1-\mu & \mu & \mu & 0 \\ \mu & 1-\mu & \mu & 0 \\ \mu & \mu & 1-\mu & 0 \\ 0 & 0 & 0 & (1-2\mu)/2 \end{bmatrix}$$

where μ is Poisson's ratio and E is Young's modulus.

To determine $\mathbf{K}_{uu}$ the product $\mathbf{B}^t\mathbf{EB}$ is integrated over the

FINITE ELEMENT PRINCIPLES

area of the triangular element. However when the element is small a good approximation is to assume constant strain and replace the radial co-ordinate, r, in **B**, by a fixed radius, say R_0, equal to the radius to the centroid of the element's area. Under this approximation:-

$$\mathbf{K}_{uu} = 2\pi R_0 \mathbf{B}^t \mathbf{E}\mathbf{B} \int_A drdz = 2\pi R_0 \mathbf{B}^t \mathbf{E}\mathbf{B}A ,$$

where A is the area of the triangle.

In a similar manner the electrical potential is discretised and approximated using linear shape functions, $\mathbf{N}_\phi$, and scalar nodal-potentials, ϕ_1, ϕ_2, ϕ_3.

5.1 Thin-walled cylinder

To illustrate the above procedure consider a thin-walled piezoceramic cylindrical element of length, h, wall thickness, t and radius, R, which is subject to a uniform circumferential static-pressure, P_0, as shown in Fig.4. The cylinder has end electrodes and is polarised along its axial direction, z. The element has two nodes with displacements and potentials given by:-

$$\mathbf{a} = [a_1, b_1, a_2, b_2]^t \quad \text{and} \quad \boldsymbol{\phi} = [\phi_1, \phi_2]^t .$$

Under uniform radial loading $b_1 = b_2 = b$ and therefore:-

$$u = (1-z/h)a_1 + (z/h)a_2$$
and
$$v = b$$

These can be written as:-

$$\begin{bmatrix} u \\ v \end{bmatrix} = \begin{bmatrix} (1-z/h) & 0 & z/h & 0 \\ 0 & 1/2 & 0 & 1/2 \end{bmatrix} \begin{bmatrix} a_1 \\ b \\ a_2 \\ b \end{bmatrix}$$

where the first matrix on the right hand side is recognised as the shape function $\mathbf{N}_u$.

For this case there can be no shear in the r-z plane and no strain in the radial direction, so the strains and displacements are related by:-

$$\begin{bmatrix} e_z \\ e_\theta \end{bmatrix} = \begin{bmatrix} \partial/\partial z & 0 \\ 0 & 1/r \end{bmatrix} \begin{bmatrix} u \\ v \end{bmatrix} = \mathbf{D}\mathbf{u}$$

FINITE ELEMENT PRINCIPLES

From the product of **D** and $\mathbf{N}_u$ it follows that **B** is given by:-

$$\mathbf{B} = \begin{bmatrix} -1/h & 0 & 1/h & 0 \\ 0 & 1/2R & 0 & 1/2R \end{bmatrix}$$

Also for this case the material stiffness-matrix, **E**, reduces to the planar equations:-

$$\mathbf{E} = \frac{E}{(1-\mu^2)} \begin{bmatrix} 1 & \mu \\ \mu & 1 \end{bmatrix}$$

The element elastic stiffness matrix, $\mathbf{K}_{uu}$, can now be calculated:-

$$\mathbf{K}_{uu} = \int_V \mathbf{B}^t \mathbf{E} \mathbf{B} dV = \frac{E}{(1-\mu^2)} \mathbf{C} dV = \frac{E\, 2\pi R h t \mathbf{C}}{(1-\mu^2)}$$

where **C** equals:-

$$\begin{bmatrix} 1/h^2 & -\mu/2Rh & -1/h^2 & -\mu/2Rh \\ -\mu/2Rh & 1/4R^2 & \mu/2Rh & 1/4R^2 \\ -1/h^2 & \mu/2Rh & 1/h^2 & \mu/2Rh \\ -\mu/2Rh & 1/4R^2 & \mu/2Rh & 1/4R^2 \end{bmatrix}$$

To calculate the remaining stiffness matrices in equations (4.3a) and (4.3b) it is necessary to consider the electrical potential along the z-direction. This is approximated by linear interpolation functions:-

$$[\ (1-z/h)\ ,\ z/h\ ] \begin{bmatrix} \phi_1 \\ \phi_2 \end{bmatrix} = \mathbf{N}_\phi \begin{bmatrix} \phi_1 \\ \phi_2 \end{bmatrix}$$

where $\mathbf{N}_\phi$ is the shape function for potential.

In the absence of shear only the z-component of electric field strength, $\mathcal{E}_z$ is non-zero. This is related to the potential by:-

$$\mathcal{E}_z = -\partial\phi/\partial z = -\partial/\partial z\ (\,[\,(1-z/h)\ ,\ z/h]\begin{bmatrix} \phi_1 \\ \phi_2 \end{bmatrix}\,)$$

$$= - [-1/h \ , \ 1/h] \begin{bmatrix} \phi_1 \\ \phi_2 \end{bmatrix}$$

where $D_e = [-1/h \ , \ 1/h]$.

The piezoelectric equations, (4.1), for this case become:-

$$\begin{bmatrix} \sigma_z \\ \sigma_\theta \end{bmatrix} = \frac{E}{(1-\mu^2)} \begin{bmatrix} 1 & \mu \\ \mu & 1 \end{bmatrix} \begin{bmatrix} e_z \\ e_\theta \end{bmatrix} - \mathcal{E}_z \begin{bmatrix} e_l \\ e_t \end{bmatrix}$$

and $$q_z = [e_l \ , \ e_t] \begin{bmatrix} e_z \\ e_\theta \end{bmatrix} + \epsilon \mathcal{E}_z$$

where the piezoelectric constants, e_{zz} and $e_{z\theta}$, which are equivalent to the more conventional e_{33} and e_{13}, have been written for convenience as e_l and e_t respectively. Also the matrix e_p therefore becomes:-

$$e_p = [e_l \ , \ e_t]^t$$

The mixed and electric stiffness matrices, $\mathbf{K}_{u\phi}$, $\mathbf{K}_{\phi u}$ and $\mathbf{K}_{\phi\phi}$ may now be evaluated using matrices $\mathbf{B}$, $\mathbf{D}_e$ and e_p and the scalar permittivity, ϵ, using the definitions associated with equations (4.3). These evaluations are straightforward and the results when substituted into (4.3) give the following pair of matrix equations:-

$$\begin{bmatrix} F_{z1} \\ F_{R1} \\ F_{z2} \\ F_{R2} \end{bmatrix} = \frac{2\pi RthE\ \mathbf{C}}{(1-\mu^2)} \begin{bmatrix} a_1 \\ b \\ a_2 \\ b \end{bmatrix} + 2\pi Rt \begin{bmatrix} e_l/h & -e_l/h \\ -e_t/2R & e_t/2R \\ -e_l/h & e_l/h \\ -e_t/2R & e_t/2R \end{bmatrix} \begin{bmatrix} \phi_1 \\ \phi_2 \end{bmatrix}$$

and $$\begin{bmatrix} Q_1 \\ Q_2 \end{bmatrix} = -\mathbf{K}_{u\phi}^t \begin{bmatrix} a_1 \\ b \\ a_2 \\ b \end{bmatrix} + 2\pi Rt\epsilon/h \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} \phi_1 \\ \phi_2 \end{bmatrix}$$

where it is to be noted that $\mathbf{K}_{\phi u} = \mathbf{K}_{u\phi}^t$.

These two equations are the piezoelectric element equations for this thin-walled end-electroded cylinder under static conditions. The hydrostatic receiving sensitivity for this case can be found by simply considering a single element.

5.2 Receiving sensitivity

Consider the electrical terminals are open-circuit such that Q_1 and Q_2 are zero and let the ends be shielded from the hydrostatic pressure such that $F_{z1} = F_{z2} = 0$. Because of the symmetry $a_1 = -a_2 = -a$ and since the pressure is uniform, then the total radial force is:-

$$-2\pi RhP_0$$

This may be considered to be equally divided between the two nodes, such that the applied nodal forces become:-

$$F_{R1} = F_{R2} = -\pi RhP_0$$

Let the potential at node 1 be set to zero so that the potential at node 2 becomes the received potential, ie. $\phi_1 = 0$.

With these various conditions substituted into the element matrix equations the following three equations result:-

$$0 = 2\pi RhtE_p(-2a/h^2 - \mu b/Rh) - 2\pi Rte_1\phi_2/h$$

$$-\pi RhP_0 = 2\pi RhtE_p(\mu a/Rh + b/2R^2) + 2\pi Rte_t\phi_2/2R$$

$$0 = 2\pi Rt(-2e_1 a/h - e_t b/R) + 2\pi Rt\epsilon\phi_2/h$$

where $E_p = E/(1-\mu^2)$.
These equations when solved give the following value for ϕ_2:-

$$\phi_2 = RhP_0 g_t/t ,$$

ie. the receiving sensitivity, $M_0 = \phi_2/P_0$, becomes the standard result:-

$$M_0 = Rhg_t/t ,$$

where the piezoelectric constant g_t is given by:-

$$g_t = (\mu e_1 - e_t)/[E\epsilon + e_1(e_1 - \mu e_t) + e_t(e_t - \mu e_1)].$$

6.0 DISCUSSION

The principles of FEM have been given above but there are a few further comments that need to be made; some of these are elaborated upon in companion papers in this proceedings.

The discretisation process, whilst easily imagined never the less does involve some complexity in deciding upon the best fitting procedure for the elements in the domain. For example a finer mesh structure is likely to be required, in regions in the domain where high displacement gradients are present, than is to be required in regions where low displacement gradients are to be found. Also, the fitting of regularly shaped elements to curved boundaries may not be optimum and so isoparametric elements, which themselves have curved boundaries, are sometimes used.

In the one- and two-dimensional examples considered in this paper the displacement matrix had either one or two components for each node, whereas in three-dimensional applications there will in general be three displacement components at each node. This means that the components of the stiffness matrix, eg. E_1 and E_2 in the example of Section 3.0, will become (3 X 3) matrices. Therefore although the assembly process proceeds in exactly the same way as described here, never the less proper account must be taken of the component matrices when considering storage requirements.

For elements which use nodes just at the corners, eg. at the three vertices of a triangle, and provided only displacement continuity is required across element boundaries, then simple linear shape functions may be used. However, if there are nodes at other positions, eg. on the perimeter at the mid-points between nodes, then the shape functions need to be higher order polynomials.

In computing an element's stiffness matrix an integration is involved, which for the case of the simple examples was performed analytically. However, where the shape functions are more complicated and the elements may have many orientations it may be necessary to carry out the integrations numerically.

Steady-state dynamics have been considered throughout, but it is possible to retain the time as a discretised parameter and use FEM to evaluate the time history or transient behaviour of the displacement or potential.

There is a considerable ammount of computation normally required when FEM is applied to a practical problem. As a result of the build up in round-off errors the overall accuracy of the results may be impaired; consideration of this problem is therefore important. A companion paper in this proceedings addresses this.

FINITE ELEMENT PRINCIPLES

Finally in the case of transducer applications it is often necessary to include the effect of fluid loading. This is accomplished by the use of some form of boundary elements to represent the fluid, but there are problems because of the usual unbounded nature of the fluid and the difficulties associated with trying to adequately represent this extended fluid by a layer of boundary elements. Another consequence of this is that sound radiation into the fluid produces damping which results in the arithmetic becoming complex rather than real. This also occurs if the element equations themselves include frictional damping terms.

7.0 CONCLUSIONS

This paper has attempted to outline the principles behind the Finite Element Method by concentrating on two relatively simple problems, where the element equations are easily obtained and the process of assembly is easily demonstrated. In a paper of this length it is not possible to cover the subject in greater depth and so if further details on the basics are required the following bibliography, which was consulted by the present author in preparing this paper, should be consulted. No attempt has been made to examine the computing implications of FEM since this is covered explicitly by other papers within this proceedings.

8.0 BIBLIOGRAPHY

O. C. Zienkiewicz, "The Finite Element Method", McGraw-Hill, 1977.

R. H. Gallagher, "Finite Element Analysis, Fundamentals", Prentice-Hall, Inc., 1975.

A. J. Davies, "The Finite Element Method: A First Approach", Clarendon Press, Oxford, 1980.

H. Allik and T. Hughes, "Finite Element Method for Piezoelectric Vibration", Int. J. Numer. Method. Eng. **2**, 151-157, 1970.

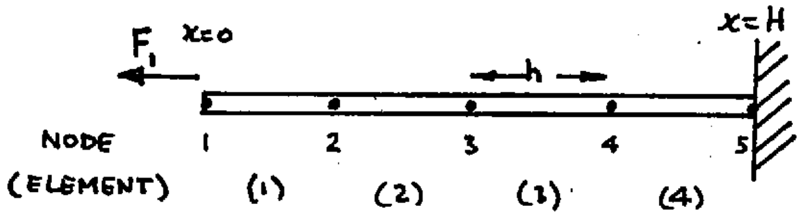

Fig.1 Geometry for one-dimensional elastic bar.

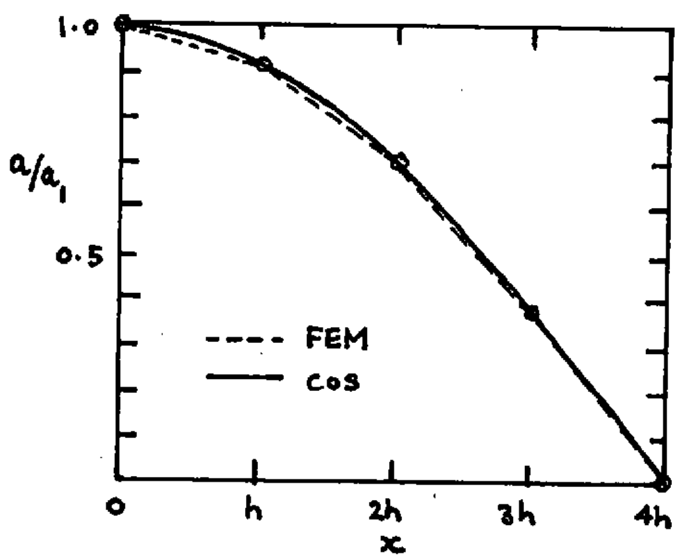

Fig.2 Normalised nodal displacements for the elastic bar

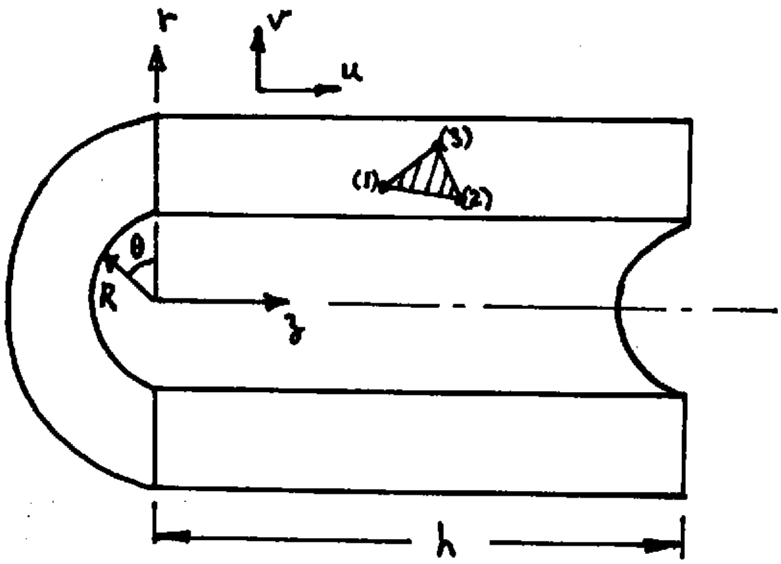

Fig.3 Cross-section through a cylinder showing a triangular element and the geometry.

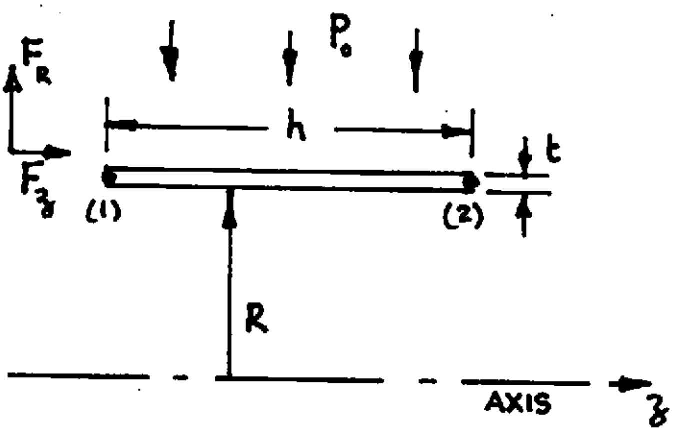

Fig.4 Finite element for a thin-walled cylinder