Chapter 7 - Simple Model for Finite Model Analysis

The Concept Behind FEA

The finite element method was originally developed to solve problems in the analysis of structures. Therefore, it is natural to use a solid mechanics example to explain the process. In FEM the loaded structure is modeled with a mesh of separate elements (Fig. 1). We shall use triangular elements here for simplicity, but later we shall discuss other important elements of various shapes. The distribution of elements is called a *mesh*, and the connecting points are called *nodes*. For stress analysis, a solution is arrived at by using basic solid mechanics equations to compute the strain from the displacement of the nodal points due to the forces and moments at the nodes. The stress is determined with the appropriate stress-strain relationship or constitutive equation. However, the problem is more complex than first seen, because the force at each node depends on the force at every other node. The elements behave like a system of springs and deform until all forces are in equilibrium. That leads to a complex system of simultaneous equations. Matrix algebra is needed to handle the cumbersome systems of equations. The key piece of information is the *stiffness matrix* for each element. It can be thought of as a spring constant that can be used to describe how much the nodal points are displaced under a system of applied forces.

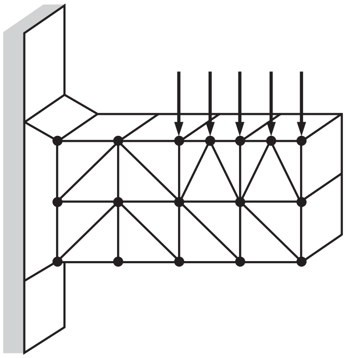

FIGURE 1
Simple finite element representation of a beam.

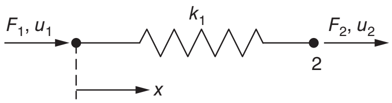

FIGURE 2
Model for a single linear element.

The simplest of all elements is a linear one-dimensional element that only supports an axial load, We model this as a spring, with a node at each end of the element, Fig. 2. The force F at a node may be related to the displacement u at the same node by the equations

$$F_1 = k_1(u_1 - u_2) = k_1u_1 - k_1u_2 \tag{1}$$

$$F_2 = k_1(u_2 - u_1) = -k_1u_1 + k_1u_2 \tag{2}$$

These equations can be written in matrix form as

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \tag{3}$$

The 2 × 2 matrix is the *stiffness matrix* for the linear element. An important property of the stiffness matrix is that it is a symmetric matrix, that is, $k_{ij} = k_{ji}$

A numerical solution for the linear (axial) element can be obtained from the cross-sectional area of the element A and its elastic modulus, E. From the definition of stress σ and Young's modulus E:

$$\sigma = Ee; \quad \frac{F_1}{A_1} = E\frac{u_1}{L_1}; \quad F_1 = \frac{EA_1u_1}{L_1} = k_1u_1$$

Therefore, the stiffness of the element is $k_1 = \frac{F_1}{u_1} = \frac{EA_1}{L_1}$ (4)

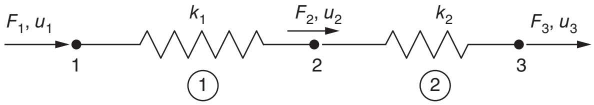

FIGURE 3
Model for two linear elements in series.

Now we expand this concept to the consideration of an axial loaded structure consisting of two linear elements, Fig. 3. Using Eq. (103.27), the force-displacement equation for each element may be written as:

$$\text{Element 1:} \quad \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \tag{5}$$

$$\text{Element 2:} \quad \begin{Bmatrix} F_2 \\ F_3 \end{Bmatrix} = \begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} \tag{6}$$

Note that F_2, u_2 is shared by both element (1) and element (2). We need to combine all three forces into a single overall global element. To do this, expand Eqs. (5) and (6) so they include F_1, F_2, and F_3.

$$\text{Element 1:} \quad \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} = \begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \tag{7}$$

All terms on the stiffness matrix that contain a subscript 3 are zero, since element 1 does not interact with node 3.

$$\text{Element 2:} \quad \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} = \begin{bmatrix} 0 & 0 & 0 \\ 0 & k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \tag{8}$$

The global stiffness matrix is obtained by adding Eqs. (8) and (9), term by term.

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} = \begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \tag{9}$$

A rule of thumb for constructing the global matrix is that if nodes m and n are not connected by an element, then $k_{mn} = 0$. For example, there is no connection between nodes (1) and (3), so $k_{31} = k_{13} = 0$.

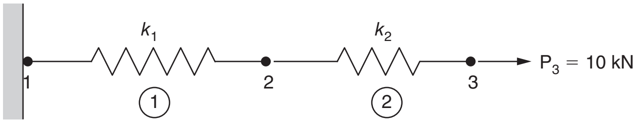

FIGURE 4
Two-element model of example problem.

EXAMPLE

Two bars of different material are welded together, end-to-end. The left end of the structure is firmly attached to a wall, and the right end is axially loaded with a force of 10 kN. The properties of the bars are:

Bar (1)	Bar (2)
Mild steel	Aluminum alloy
$A_1 = 70\ \text{mm}^2$	$A_2 = 70\ \text{mm}^2$
$L_1 = 100\ \text{mm}$	$L_2 = 280\ \text{mm}$
$E_1 = 200\ \text{GN/m}^2$	$E_2 = 70\ \text{GN/m}^2$

Find the stress in each bar and the total elongation of the structure. Also find the reaction force of the structure on the wall.

Start by modeling the problem with linear elastic elements, Fig. 4. Note that because of the uniform geometry, the element (1) is the entire steel bar, and element (2) is the aluminum alloy bar.

Find the spring constant for each element and construct the stiffness matrix.

$$k_1 = \frac{A_1E_1}{L_1} = \frac{(70)200\times10^3}{100} = 140\,\text{kN/mm};\ k_2 = \frac{A_2E_2}{L_2} = \frac{(70)\times70\times10^3}{280} = 17.5\text{kN/mm}$$

The global stiffness matrix from Eq (10.34) is $K = \begin{bmatrix} 140 & -140 & 0 \\ -140 & 157.5 & 17.5 \\ 0 & -17.5 & 17.5 \end{bmatrix}$

Also, from Eq. (9),

$$\begin{Bmatrix} F_1 \\ 0 \\ 10 \end{Bmatrix} = \begin{bmatrix} 140 & -140 & 0 \\ -140 & 157.5 & -17.5 \\ 0 & -17.5 & 17.5 \end{bmatrix} \begin{Bmatrix} 0 \\ u_2 \\ u_3 \end{Bmatrix}$$ where, from the boundary conditions at the wall, $u_1 = 0$.

From the matrix equation we write the three linear algebraic equations representing the two-element system.

$$\begin{aligned} P_1 &= 140(0) - 140u_2 + (0)u_3 \\ 0 &= -140(0) + 157.5u_2 - 17.5u_3 \\ 10 &= 0 - 17.5u_2 + 17.5u_3 \end{aligned}$$

The bottom two equations can be solved simultaneously to give $u_2 = 0.0714$ mm and $u_3 = 0.6390$ mm. The total elongation of the structure is $0.0714 + 0.6390 = 0.7104$ mm.

Then from the first equation, $P_1 = -140(0.0714) = 9.996$ kN.
The structure pushes on the wall with a force of $9.996 \approx 10$ kN.
(Note: we could have obtained this from the summation of forces in the *x* direction.)

Now we need to find the stresses in each bar. The stress in bar 1 is

$$\sigma_{(1)} = E_1 e_1 = E_1\left(\frac{u_2 - u_1}{L_1}\right) = 200\times 10^9 \frac{N}{m^2}\times\frac{1}{10^6}\frac{m^2}{mm^2}\left(\frac{0.0714 - 0.0}{100}\right)\frac{mm}{mm}$$

$$= 200\times 10^3(7.104\times 10^{-4}) = 1420\frac{N}{mm^2} = 1420\frac{N}{mm^2} = 142\,\text{MPa} = 20{,}600\,\text{psi}$$

$$\sigma_{(2)} = E_2 e_2 = E_2\left(\frac{u_3 - u_2}{L_2}\right) = 70\times 10^3\left(\frac{0.6390 - 0.0714}{280}\right) = 142\,\text{MPa} = 20{,}600\,\text{psi}$$

Note that both elements (bars) have the same axial stress even though their materials have a different elastic modulus. Physically, this is what we should expect from the simple equation $\sigma = P/A$ since the load on each bar and its area is the same.

This section discusses the most elementary FEA element possible, an axial linear element with only a single degree of freedom (DOF). From what you learned during the first week in your mechanics of materials course, you would have quickly found the stresses in the structure without resorting to the complexity of the matrix equations. However, if we used a three-dimensional beam element there are 6 DOF, and the possibility of moments and forces normal to the axes, which are not possible with a linear element. The mathematics quickly becomes very complex, and computer numerical analysis becomes a must.