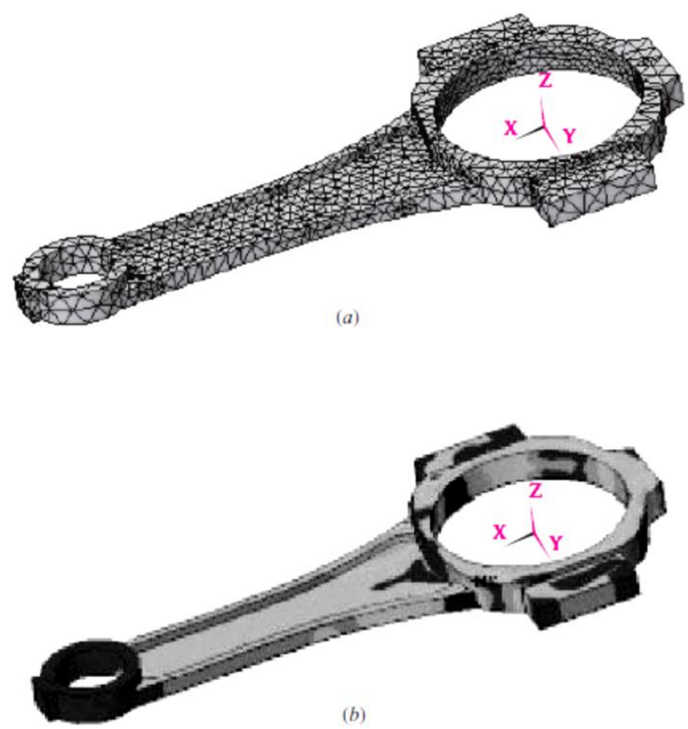
Finite-Element Analysis

Finite-element analysis is one of the most important and is easily integrated into the computer-aided engineering environment. Solid-modeling CAD software provides an excellent platform for the easy creation of FEA models.

- Mechanical components in the form of simple bars, beams, etc., can be analyzed quite easily by basic methods of mechanics that provide closed-form solutions.
- There are a great many numerical techniques used in engineering applications for which the digital computer is very useful. In mechanical design, where computer-aided design (CAD) software is heavily employed, the analysis method that integrates well with CAD is finite-element analysis (FEA).
- There is also a number of commercial FEA software packages that are available, such as ANSYS, NASTRAN, Algor, etc.
- There are a multitude of FEA applications such as static and dynamic, linear and nonlinear, stress and deflection analysis; free and forced vibrations; heat transfer (which can be combined with stress and deflection analysis to provide thermally induced stresses and deflections); elastic instability (buckling); acoustics; electrostatics and magnetics (which can be combined with heat transfer); fluid dynamics; piping analysis; and multiphysics.

Figure 19–1

Model of a crankshaft using ANSYS finite-element software. (*a*) Meshed model (*b*); stress contours. *Courtesy of S. Boedo (see footnote 1).*



❖ The Finite-Element Method

The finite-element method is a numerical technique that discretizes the domain of a continuous structure. For an example of discretization errors, consider the constant thickness, thin plate structure shown in Fig. 19–2a. Figure 19–2b shows a finite-element model of the structure where three-node, plane stress, simplex triangular elements are

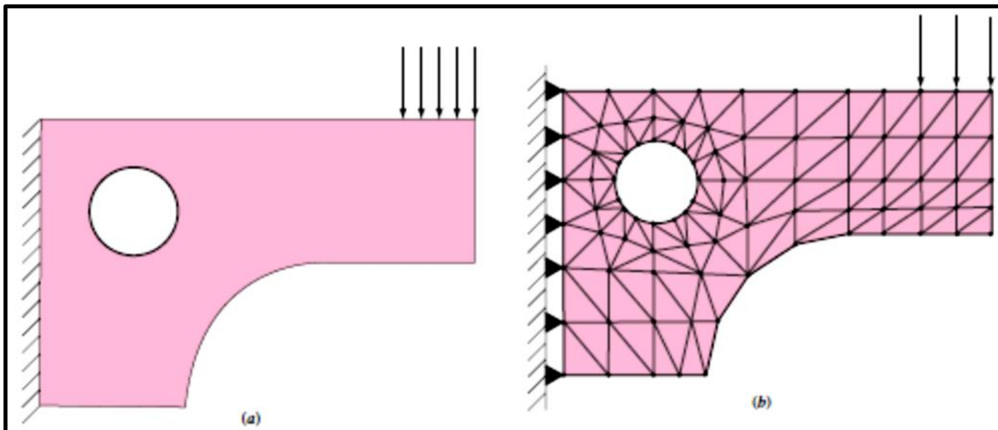


Figure 19–2
Structural problem. (*a*) Idealized model; (*b*) finite-element model.

employed. This element type has a flaw that creates two basic problems. The element has straight sides that remain straight after deformation. The strains throughout the plane stress triangular element are constant.

- The first problem, a geometric one, is the modeling of curved edges. Note that the surface of the model with a large curvature appears poorly modeled, whereas the surface of the hole seems to be reasonably modeled.
- The second problem, is that the strains in various regions of the actual structure are changing rapidly, and the constant strain element will provide only an approximation of the average strain at the center of the element. The results can be improved by significantly increasing the number of elements (increased mesh density). Alternatively, using a better element, such as an eight-node quadrilateral, which is more suited to the application, will provide the improved results. Because of higher-order interpolation functions, the eight-node quadrilateral element can model curved edges and provide for a higher-order function for the strain distribution.

In Fig. 19–2*b*, the triangular elements are shaded and the nodes of the elements are represented by the black dots. Forces and constraints can be placed only at the nodes.

The nodes of a simplex triangular plane stress elements have only two degrees of freedom, translation in the plane. Thus, the solid black, simple support triangles on the left edge represent the fixed support of the model. Also, the distributed load can be applied only to three nodes as shown. The modeled load has to be statically consistent with the actual load.

❖ Finite-Element Analysis Steps

- An actual mechanical component is a continuous elastic structure (continuum).
- FEA divides (discretizes) the structure into small but finite, well-defined, elastic substructures (elements).
- By using polynomial functions, together with matrix operations, the continuous elastic behavior of each element is developed in terms of the element's material and geometric properties.
- Loads can be applied within the element (gravity, dynamic, thermal, etc.), on the surface of the element, or at the *nodes* of the element.
- The element's nodes are the fundamental governing entities of the element, as it is the node where the element connects to other elements, where elastic properties of the element are eventually established, where boundary conditions are assigned, and where forces (contact or body) are ultimately applied.
- A node possesses *degrees of freedom* (dof's). Degrees of freedom are the independent translational and rotational motions that can exist at a node. At most, a node can possess three translational and three rotational degrees of freedom.
- Once each element within a structure is defined *locally* in matrix form, the elements are then *globally* assembled (attached) through their common nodes (dof's) into an overall system matrix.
- Applied loads and boundary conditions are then specified and through matrix operations the values of all unknown displacement degrees of freedom are determined.
- Once this is done, it is a simple matter to use these displacements to determine strains and stresses through the constitutive equations of elasticity.

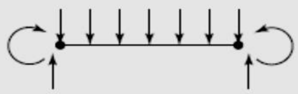
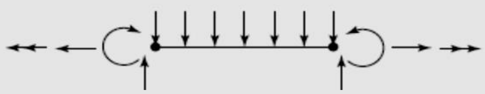
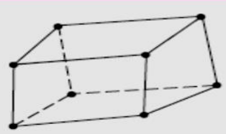
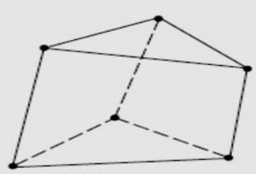
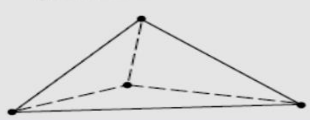
❖ Element Geometries

Many geometric shapes of elements are used in finite-element analysis for specific applications. The various elements used in a general-purpose commercial FEM software

code constitute what is referred to as the element library of the code. Elements can be placed in the following categories: line elements, surface elements, solid elements, and special-purpose elements. Table 19–1 provides some, but not all, of the types of

Table 19–1

Sample finite-element library.

Element Type	None	Shape	Number of Nodes	Applications
Line	Truss		2	Pin-ended bar in tension or compression
	Beam		2	Bending
	Frame		2	Axial, torsional, and bending. With or without load stiffening.
Surface	4-node quadri-lateral		4	Plane stress or strain, axisymmetry, shear panel, thin flat plate in bending
	8-node quadri-lateral		8	Plane stress or strain, thin plate or shell in bending
	3-node triangular		3	Plane stress or strain, axisymmetry, shear panel, thin flat plate in bending. Prefer quad where possible. Used for transitions of quads.
	6-node Triangular		6	Plane stress or strain, axisymmetry, thin plate or shell in bending. Prefer quad where possible. Used for transitions of quads.
Solid†	8-node hexagonal (brick)		8	Solid, thick plate
	6-node pentagonal (wedge)		6	Solid, thick plate. Used for transitions.
	4-node tetrahedron (tet)		4	Solid, thick plate. Used for transitions.
Special purpose	Gap		2	Free displacement for prescribed compressive gap
	Hook		2	Free displacement for prescribed extension gap
	Rigid		Vari-able	Rigid constraints between nodes

†These elments are also available with midside nodes.

elements available for finite-element analysis for structural problems.

Not all elements support all degrees of freedom. For example, the 3-D truss element supports only three translational degrees of freedom at each node.

❖ The Finite-Element Solution Process

- We will describe the finite-element solution process on a very simple one-dimensional problem, using the linear truss element bar loaded in tension or compression and is of constant cross-sectional area A, length l, and elastic modulus E.
- The basic truss element has two nodes, and for a one-dimensional problem, each node will have only one degree of freedom.
- A truss element can be modeled as a simple linear spring.

$$k = \frac{AE}{l} \qquad (19\text{–}1)$$

- Consider a spring element (e) of spring rate k_e, with nodes i and j, as shown in Fig. 19–3. Nodes and elements will be numbered.

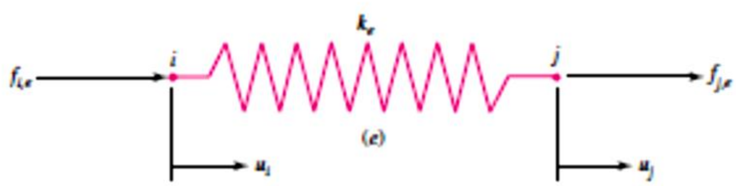


Figure 19–3

A simple spring element.

- Assuming all forces f and displacements u directed toward the right as positive, the forces at each node can be written as

$$\begin{aligned} f_{i,e} &= k_e\left(u_i - u_j\right) = k_e u_i - k_e u_j \\ f_{j,e} &= k_e\left(u_j - u_i\right) = -k_e u_i + k_e u_j \end{aligned} \qquad (19\text{–}2)$$

The two equations can be written in matrix form as

$$\begin{Bmatrix} f_{i,e} \\ f_{i,e} \end{Bmatrix} = \begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} \qquad (19\text{–}3)$$

- Now, consider a two-spring system as shown in Fig. 19–4a. Here we have numbered the nodes and elements. We have also labeled the forces at each node. However, these forces are the total external forces at each node, F_1, F_2, and F_3. If we draw separate free-body diagrams we will expose the internal forces as shown in Fig. 19–4b.

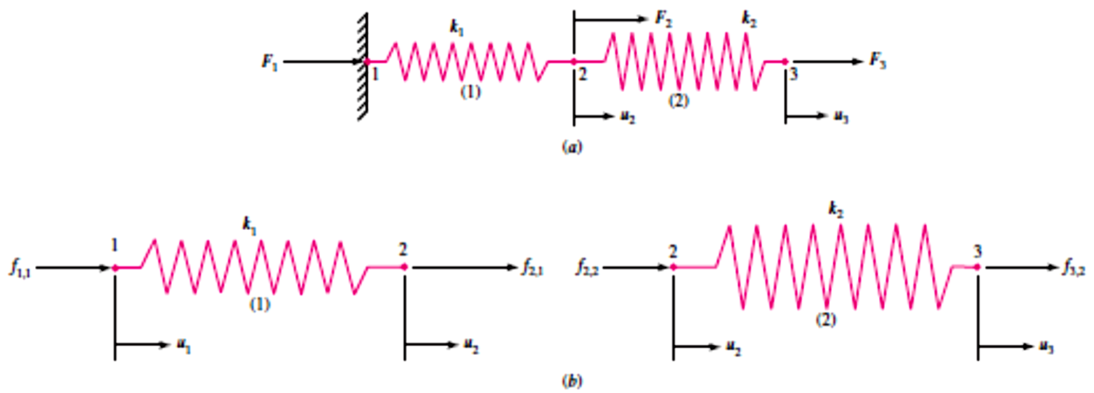


Figure 19–4
A two-element spring system. (a) System model, (b) separate free-body diagrams.

Using Eq. (19–3) for each spring gives

Element 1	$\begin{Bmatrix} f_{1,1} \\ f_{2,1} \end{Bmatrix} = \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$	(19–4a)
Element 2	$\begin{Bmatrix} f_{2,2} \\ f_{3,2} \end{Bmatrix} = \begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix}$	(19–4b)

The total force at each node is the external force, $F_1 = f_{1,1}$, $F_2 = f_{2,1} + f_{2,2}$, and $F_3 = f_{3,2}$. Combining the two matrices in terms of the external forces gives

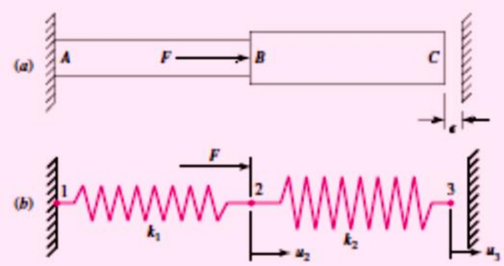
$$\begin{Bmatrix} f_{1,1} \\ f_{2,1} + f_{2,2} \\ f_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} = \begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & (k_1 + k_2) & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \qquad (19\text{–}5)$$

- If we know the displacement of a node, then the force at the node will be unknown. For example, in Fig. 19–4*a*, the displacement of node 1 at the wall is zero, so *F*1 is the unknown reaction force.
- For example, in Fig. 19–4a, the displacements at nodes 2 and 3 are unknown, and the forces F2 and F3 are to be specified.

EXAMPLE 19-1 Consider the aluminum step-shaft shown in Fig. 19–5*a*. The areas of sections *AB* and *BC* are 0.100 in^2 and 0.150 in^2, respectively. The lengths of sections *AB* and *BC* are 10 in and 12 in, respectively. A force $F = 1000$ lbf is applied to *B*. Initially, a gap of $\epsilon = 0.002$ in exists between end *C* and the right rigid wall. Determine the wall reactions, the internal forces in the members, and the deflection of point *B*. Let $E = 10$ Mpsi and assume that end *C* hits the wall. Check the validity of the assumption.

Figure 19–5

(*a*) Step shaft, (*b*) spring model.



Solution The step-shaft is modeled by the two-spring system of Fig. 19–5*b* where

$$k_1 = \left(\frac{AE}{l}\right)_{AB} = \frac{0.1\,(10)\,10^6}{10} = 1\,(10^5)\ \text{lbf/in}$$

$$k_2 = \left(\frac{AE}{l}\right)_{BC} = \frac{0.15\,(10)\,10^6}{12} = 1.25\,(10^5)\ \text{lbf/in}$$

With $u_1 = 0$, $F_2 = 1000$ lbf and the assumption that $u_3 = \epsilon = 0.002$ in, Eq. (19.5) becomes

$$\begin{Bmatrix} F_1 \\ 1000 \\ F_3 \end{Bmatrix} = 10^5 \begin{bmatrix} 1 & -1 & 0 \\ -1 & 2.25 & -1.25 \\ 0 & -1.25 & 1.25 \end{bmatrix} \begin{Bmatrix} 0 \\ u_2 \\ 0.002 \end{Bmatrix} \tag{1}$$

For large problems, there is a systematic method of solving equations like Eq. (1), called *partitioning* or *the elimination approach*.[11] However, for this simple problem, the solution is quite simple. From the second equation of the matrix equation

$$1000 = 10^5[-1(0) + 2.25\,u_2 - 1.25(0.002)]$$

or,

Answer
$$u_B = u_2 = \frac{1000/10^5 + 1.25\,(0.002)}{2.25} = 5.556\,(10^{-3})\ \text{in}$$

Since $u_B > \epsilon$, it is verified that point *C* hits the wall.

The reactions at the walls are F_1 and F_3. From the first and third equations of matrix Eq. (1),

Answer
$$F_1 = 10^5[-1(u_2)] = 10^5[-1(5.556)10^{-3}] = -555.6\ \text{lbf}$$

and

Answer

$$F_3 = 10^5[-1.25u_2 + 1.25(0.002)]$$
$$= 10^5[-1.25(5.556)10^{-3} + 1.25(0.002)] = -444.4 \text{ lbf}$$

Since F_3 is negative, this also verifies that C hits the wall. Note that $F_1 + F_3 = -555.6 - 444.4 = -1000$ lbf, balancing the applied force (with no statics equations necessary).

For internal forces, it is necessary to return to the individual (local) equations. From Eq. (19–4*a*),

$$\begin{Bmatrix} f_{1,1} \\ f_{2,1} \end{Bmatrix} = \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = 10^5 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} 0 \\ 5.556(10^{-3}) \end{Bmatrix} = \begin{Bmatrix} -555.6 \\ 555.6 \end{Bmatrix} \text{lbf}$$

Answer Since $f_{1,1}$ is directed to the left and $f_{2,1}$ is directed to the right, the element is in tension, with a force of 555.6 lbf. If the stress is desired, it is simply $\sigma_{AB} = f_{2,1}/A_{AB} = 555.6/0.1 = 5556$ psi.

For element *BC*, from Eq. (19.4*b*),

$$\begin{Bmatrix} f_{2,2} \\ f_{3,2} \end{Bmatrix} = \begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = 10^5 \begin{bmatrix} 1.25 & -1.25 \\ -1.25 & 1.25 \end{bmatrix} \begin{Bmatrix} 5.556(10^{-3}) \\ 0.002 \end{Bmatrix} = \begin{Bmatrix} 444.5 \\ -444.5 \end{Bmatrix} \text{lbf}$$

Answer Since $f_{2,2}$ is directed to the right and $f_{3,2}$ is directed to the left, the element is in compression, with a force of 444.5 lbf. If the stress is desired, it is simply $\sigma_{BC} = -f_{2,2}/A_{BC} = -444.5/0.15 = -2963$ psi.