Type of Finite Elements and Steps in FEA Process

Types of Elements

Finite element analysis was originally developed for two-dimensional (plane stress) situations. A three-dimensional solid causes an orders-of-magnitude increase in the number of simultaneous equations that must be solved, but by using higher-order mesh elements and faster computers these issues are routinely handled in FEA. Broadly, a *continuum element* is one whose geometry is completely defined by the coordinates of the nodes and allows variation of the displacement based on the deformation of the elements. Figure 2 shows a few of the elements available in FEA. Triangles and quadrilaterials, Fig. 2*a* and *b*, are the simplest plane elements, with two degrees of freedom at each node. Adding additional nodes, either at the centroid or along the edges (Fig. 2*c*), provides for curved edges and faces. Whenever the boundaries are curved in three dimensions, a special class of elements called *isoparametric elements* are used. Figure 2*d* is an isoparametric triangle, (e) is a tetrahedron (tet), and (f) is a hexahedron (hex). These elements are most useful when it is desirable to approximate curved boundaries with a minimum number of elements. Another useful class of elements are *structural elements*. These are based on common structural shapes and types of loading used in the mechanics of solids. The most common structural elements are the axial element shown in Fig. 1, and in Fig. 2 the beam element (g), the plate element (h), and the shell element (i).

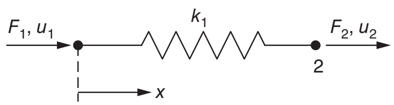

FIGURE 1
Model for a single linear element.

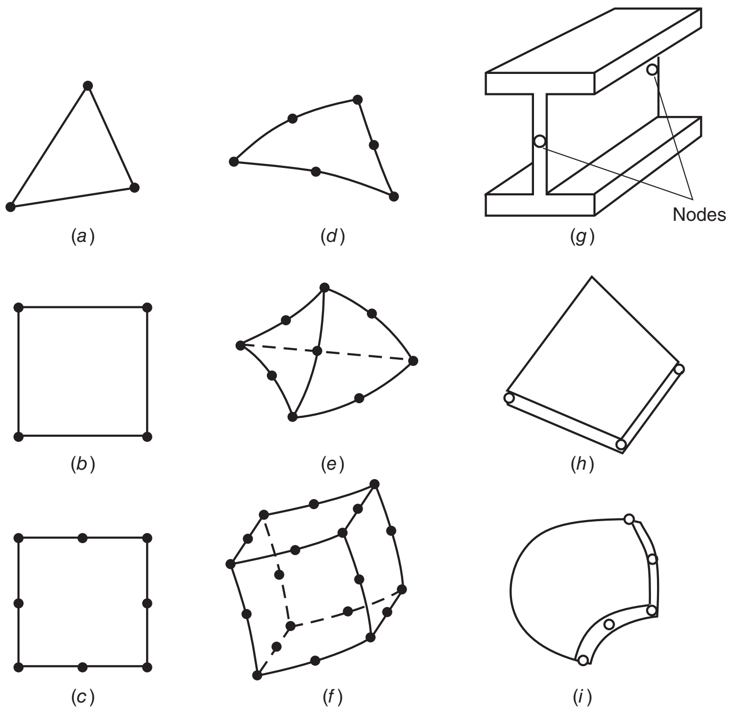

FIGURE 2
Some common elements used in finite element analysis.

With the linear axial element, finding the values of the displacement vector u was straightforward. It could be expressed as a linear polynomial $u = a_1 + a_2x$, where the constants are determined from the displacements at the nodes. In Fig. 1 , at node 1, $x = 0$, so $u = u_1 = a_1$. At node 2, $x = L$, so $u = u_2 = a_1 + a_2L$. Therefore, $a_1 = u_1$ and $a_2 = (u_2 - u_1)/L$. The variation in displacement over the element is

$$u = a_1 + a_2x = u_1 + \left(\frac{u_2 - u_1}{L}\right)x = \left(1 - \frac{x}{L}\right)u_1 + \frac{x}{L}u_2 \qquad (1)$$

This equation can be written in matrix form as

$$u = \left[1 - \frac{x}{L} \quad \frac{x}{L}\right]\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \left[\mathrm{N}\right]\left\{\mathrm{u}\right\} \qquad (2)$$

[N] is called the *shape factor* of the element. It specifies the variation in displacement within the element.

Two- and three-dimensional elements with more nodes and displacements at a node have much more complex shape factors. For example, in a 2-D triangular element with three nodes and two displacements per node, the components of displacement u along the x-axis and v along the y-axis are given by Eq. (3).

$$\begin{aligned} u &= a_1 + a_2 x + a_3 y \\ v &= a_4 + a_5 x + a_6 y \end{aligned} \qquad (3)$$

Since the strain in the x-direction is $\varepsilon_x = \dfrac{\partial u}{\partial x} = a_2$ we see that for this first-order element the strain is constant throughout the element. For a 2-D quadrilateral, with four nodes and two displacements per node, the components of the displacement are

$$\begin{aligned} u &= a_1 + a_2 x + a_3 y + a_4 xy \\ v &= a_5 + a_6 x + a_7 y + a_8 xy \end{aligned} \qquad (4)$$

Now the strain in the x-direction is $\varepsilon_x = \dfrac{\partial u}{\partial x} = a_2 + a_4 y$, which provides for a strain gradient in this higher-order element. Elements with additional nodes, like Fig. 2 , lead to still higher-order polynomials to express the strain more accurately within the element and to more accurately represent curved boundaries.

Since FEA creates a model of elements that aims to predict the behavior of a continuum, the selection of type of element and its size is very important. For example, using straight-sided triangular elements to model the region around a hole in a plate would lead to a poor approximation to the circular hole unless the size of the elements is very small. This is an example of a type of modeling error called *discretization*. *Formulation errors* arise from using elements that do not exactly duplicate the way the physical part would behave under the loading. If we think that displacements change linearly over the meshed region, then a linear element would be appropriate. However, if the displacements vary quadratically, then there would be a formulation error, and a higher-order element should be chosen. To create a good element mesh, the stress distribution, not the stresses, should be understood beforehand.

FEA software generally provides the capability for automatically meshing a solid with triangles, quadrilaterals, tetrahedrons, and hexahedrons. The accuracy of the model will be determined by its *convergence error*. This is the percent difference between the results of one run and the next iteration as either the element size or the nature of the element is changed. There are two ways that FEA software approaches this problem. In the *h-element method*, the size of the element (h is the element size) is reduced. In the *p-element method* (p for polynomial), the software increases the element's order of the polynomial describing the variation of displacement with distance without changing the original mesh.

Figure 3 illustrates these approaches. At the top left, we see the original mesh of first-order elements in an h-element approach. After the first run the analysis shows the stresses to be highest in the curved region, and the automatic mesher decreases the size of the elements in this region to account for the steep stress gradient (top right). A third iteration, with still smaller elements in the critical region, is run to see whether the stress is converging. The bottom of Fig. 3 illustrates the p-element approach. The initial mesh (bottom left) is changed by the automatic mesher to second-order

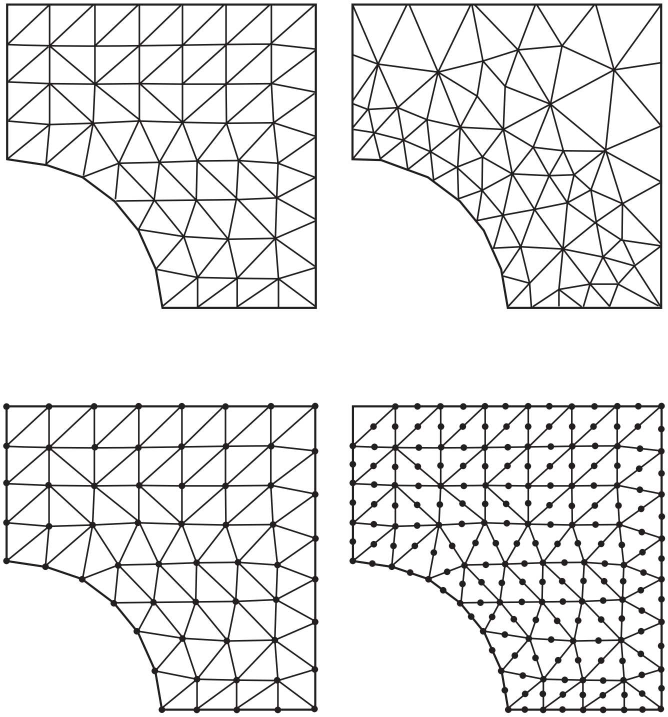

FIGURE 3
Top row—*h*-element approach to improving accuracy in remeshing. Bottom row—*p*-element approach.

elements without changing the mesh size. There is controversy as to which is the better approach. The *p*-element method gives better representation of curved surfaces and is better suited where stress gradients are high. However, it requires much greater computational resources. Moreover, if the element is too large in the *p*-element method, it can have a major impact on accuracy. Many FEA software programs provide for both approaches. As to choice of elements, most 3-D work in FEA modeling is done with the higher-order elements shown in Fig. 2 *e* and *f*. The 10-node tetrahedron element and the 20-node hexahedron element provide good results for stress analysis at reasonable mesh size with a similar number of nodes.[14] However, the 10-node tetrahedron elements provide accuracy comparable to the 20-node hexahedron at less computation time.

Steps in FEA Process

Finite element modeling is divided into three phases: preprocessing, computation, and post processing. However, even before entering the first phase, a careful engineer will

14. A. M. Niazy, *Machine Design,* Nov. 6, 1997, pp. 54–58.

perform a preliminary analysis to define the problem. Is the physics of the problem well enough known? What is an approximate solution based on simple methods of analysis and calculation? Does the problem really need an expensive finite element analysis?

Preprocessing phase: In the preprocessing phase the following decisions and actions are taken:

- The geometry of the part is imported from the CAD model. Because solid models contain great detail, they often must be simplified by deleting small nonstructural features and taking advantage of symmetry to reduce computation time.
- Make decisions concerning the division of the geometry into elements, often called meshing. The issue is knowing which types of elements to use, linear, quadratic, or cubic interpolation functions, and building a mesh that will provide a solution with the needed accuracy and efficiency. Most FEA software provide a means for automatically meshing the geometry. The finite element mesh is applied in one of two ways: structured (mapped) mesh or unstructured (free) mesh. Structured meshes have a clear structure of triangles or quadrilateral elements (for 2-D) or tets or hexes (for 3-D) that are produced by rule-based mapping techniques. Grid points can be distributed along lines with effective spacing, and well-graded grids can be constructed. This approach is effective when the geometry is relatively simple. With complex geometries a multiblock approach is used, in which the geometry is filled with an assemblage of meshed cubes. This requires the additional step of setting up the connections between the blocks. Unstructured meshing does not show structure in the placement of the elements.
- Determine how the structure is loaded and supported, or in a thermal problem determine the initial conditions of temperature. Make sure you understand the boundary conditions. It is important to incorporate sufficient restraints to displacement so that rigid body motion of the structure is prevented.
- Select the constitutive equation for describing the material (linear, nonlinear, etc.) that relates displacement to strain and then to stress.

Computation: The operations in this phase are performed by the FEA software.

- The FEA program renumbers the nodes in the mesh to minimize computational resources by minimizing the size of the global stiffness matrix ***K***.
- It generates a stiffness matrix *k* for each element and assembles the elements together so that continuity is maintained to form the *global* or *structural* matrix ***K***. Based on the load vector the software generates the external loads and applies displacement boundary conditions.
- Then the computer solves the massive matrix equation for the displacement vector or whatever is the dependent variable in the problem. The constraint forces are also determined.

Post processing: These operations are also performed by the FEA software.

- In a stress analysis problem, post processing takes the displacement vector and converts into strains, element by element, and then, with the appropriate constitutive equation, into a field of stress values.

- A finite element solution could easily contain thousands of field values. Therefore, post processing operations are needed to interpret the numbers efficiently. Typically the geometry of the part is shown on which contours of constant stress have been plotted . Mathematical operations may have to be performed on the data by the FEA software before it is displayed, such as determining the Von Mises effective stress.
- Increasingly, FEA software is being combined with an optimization package and used in iterative calculations to optimize a critical dimension or shape.