CHAPTER 1

Basic Concepts of the Finite Element Method

1.1 INTRODUCTION

The finite element method (FEM), sometimes referred to as *finite element analysis* (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. Boundary value problems are also sometimes called *field* problems. The field is the domain of interest and most often represents a physical structure. The *field variables* are the dependent variables of interest governed by the differential equation. The *boundary conditions* are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field. Depending on the type of physical problem being analyzed, the field variables may include physical displacement, temperature, heat flux, and fluid velocity to name only a few.

1.2 HOW DOES THE FINITE ELEMENT METHOD WORK?

The general techniques and terminology of finite element analysis will be introduced with reference to Figure 1.1. The figure depicts a volume of some material or materials having known physical properties. The volume represents the domain of a boundary value problem to be solved. For simplicity, at this point, we assume a two-dimensional case with a single field variable $\phi(x, y)$ to be determined at every point $P(x, y)$ such that a known governing equation (or equations) is satisfied exactly at every such point. Note that this implies an exact

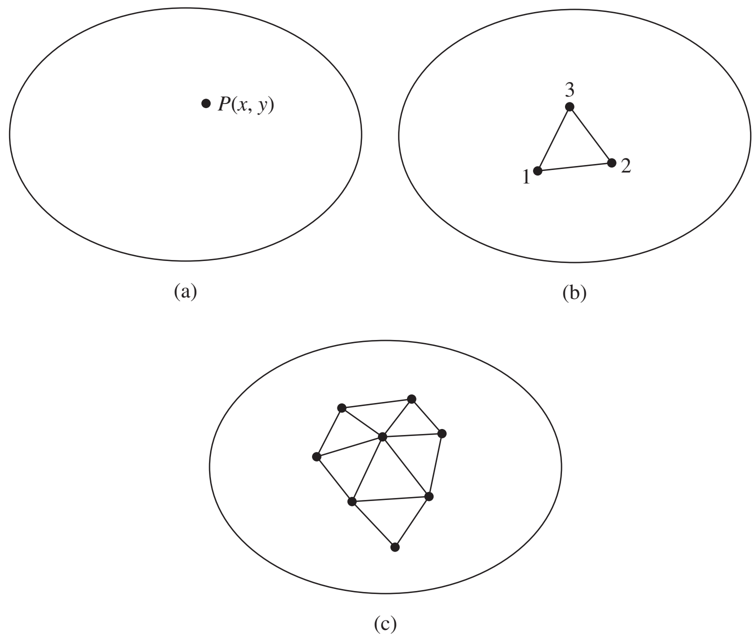


Figure 1.1
(a) A general two-dimensional domain of field variable $\phi(x, y)$.
(b) A three-node finite element defined in the domain. (c) Additional elements showing a partial finite element mesh of the domain.

mathematical solution is obtained; that is, the solution is a closed-form algebraic expression of the independent variables. In practical problems, the domain may be geometrically complex as is, often, the governing equation and the likelihood of obtaining an exact closed-form solution is very low. Therefore, approximate solutions based on numerical techniques and digital computation are most often obtained in engineering analyses of complex problems. Finite element analysis is a powerful technique for obtaining such approximate solutions with good accuracy.

A small triangular element that encloses a finite-sized subdomain of the area of interest is shown in Figure 1.1b. That this element is *not* a differential element of size $dx \times dy$ makes this a *finite element*. As we treat this example as a two-dimensional problem, it is assumed that the thickness in the z direction is constant and z dependency is not indicated in the differential equation. The vertices of the triangular element are numbered to indicate that these points are nodes. A *node* is a specific point in the finite element at which the value of the field variable is to be explicitly calculated. *Exterior* nodes are located on the boundaries of the finite element and may be used to connect an element to adjacent finite elements. Nodes that do not lie on element boundaries are *interior* nodes and cannot be connected to any other element. The triangular element of Figure 1.1b has only exterior nodes.

If the values of the field variable are computed only at nodes, how are values obtained at other points within a finite element? The answer contains the crux of the finite element method: The values of the field variable computed at the nodes are used to approximate the values at nonnodal points (that is, in the element interior) by *interpolation* of the nodal values. For the three-node triangle example, the nodes are all exterior and, at any other point within the element, the field variable is described by the approximate relation

$$\phi(x, y) = N_1(x, y)\phi_1 + N_2(x, y)\phi_2 + N_3(x, y)\phi_3 \tag{1.1}$$

where ϕ_1, ϕ_2, and ϕ_3 are the values of the field variable at the nodes, and N_1, N_2, and N_3 are the *interpolation functions,* also known as *shape functions* or *blending functions*. In the finite element approach, the nodal values of the field variable are treated as unknown *constants* that are to be determined. The interpolation functions are most often polynomial forms of the independent variables, derived to satisfy certain required conditions at the nodes. These conditions are discussed in detail in subsequent chapters. The major point to be made here is that the interpolation functions are predetermined, *known* functions of the independent variables; and these functions describe the variation of the field variable within the finite element.

The triangular element described by Equation 1.1 is said to have 3 *degrees of freedom,* as three nodal values of the field variable are required to describe the field variable everywhere in the element. This would be the case if the field variable represents a scalar field, such as temperature in a heat transfer problem (Chapter 7). If the domain of Figure 1.1 represents a thin, solid body subjected to plane stress (Chapter 9), the field variable becomes the displacement vector and the values of two components must be computed at each node. In the latter case, the three-node triangular element has 6 degrees of freedom. In general, the number of degrees of freedom associated with a finite element is equal to the product of the number of nodes and the number of values of the field variable (and possibly its derivatives) that must be computed at each node.

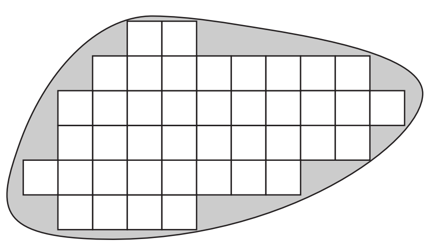
How does this element-based approach work over the entire domain of interest? As depicted in Figure 1.1c, every element is connected *at its exterior nodes* to other elements. The finite element equations are formulated such that, at the nodal connections, the value of the field variable at any connection is the same for each element connected to the node. Thus, continuity of the field variable at the nodes is ensured. In fact, finite element formulations are such that continuity of the field variable across interelement boundaries is also ensured. This feature avoids the physically unacceptable possibility of gaps or voids occurring in the domain. In structural problems, such gaps would represent physical separation of the material. In heat transfer, a "gap" would manifest itself in the form of different temperatures at the same physical point.

Although continuity of the field variable from element to element is inherent to the finite element formulation, interelement continuity of gradients (i.e., derivatives) of the field variable does not generally exist. This is a critical observation. In most cases, such derivatives are of more interest than are field variable values. For example, in structural problems, the field variable is displacement but the true interest is more often in strain and stress. As *strain* is defined in terms of first derivatives of displacement components, strain is not continuous across element boundaries. However, the magnitudes of discontinuities of derivatives can be used to assess solution accuracy and convergence as the number of elements is increased, as is illustrated by the following example.

1.2.1 Comparison of Finite Element and Exact Solutions

The process of representing a physical domain with finite elements is referred to as *meshing,* and the resulting set of elements is known as the finite element *mesh*. As most of the commonly used element geometries have straight sides, it is generally impossible to include the entire physical domain in the element mesh if the domain includes curved boundaries. Such a situation is shown in Figure 1.2a, where a curved-boundary domain is meshed (quite coarsely) using square elements. A refined mesh for the same domain is shown in Figure 1.2b, using smaller, more numerous elements of the same type. Note that the refined mesh includes significantly more of the physical domain in the finite element representation and the curved boundaries are more closely approximated. (Triangular elements could approximate the boundaries even better.)

If the interpolation functions satisfy certain mathematical requirements (Chapter 6), a finite element solution for a particular problem converges to the exact solution of the problem. That is, as the number of elements is increased and the physical dimensions of the elements are decreased, the finite element solution changes incrementally. The incremental changes decrease with the mesh refinement process and approach the exact solution asymptotically. To illustrate convergence, we consider a relatively simple problem that has a known solution. Figure 1.3a depicts a tapered, solid cylinder fixed at one end and subjected to a tensile load at the other end. Assuming the displacement at the point of load application to be of interest, a first approximation is obtained by considering the cylinder to be uniform, having a cross-sectional area equal to the average area


(a)

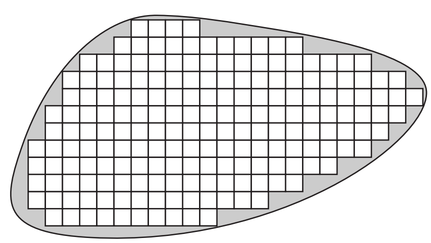

(b)

Figure 1.2
(a) Arbitrary curved-boundary domain modeled using square elements. Stippled areas are not included in the model. A total of 41 elements is shown. (b) Refined finite element mesh showing reduction of the area not included in the model. A total of 192 elements is shown.

of the cylinder (Figure 1.3b). The uniform bar is a *link* or *bar* finite element (Chapter 2), so our first approximation is a one-element, finite element model. The solution is obtained using the strength of materials theory. Next, we model the tapered cylinder as two uniform bars in series, as in Figure 1.3c. In the two-element model, each element is of length equal to half the total length of the cylinder and has a cross-sectional area equal to the average area of the corresponding half-length of the cylinder. The mesh refinement is continued using a four-element model, as in Figure 1.3d, and so on. For this simple problem, the displacement of the end of the cylinder for each of the finite element models is as shown in Figure 1.4a, where the dashed line represents the known solution. Convergence of the finite element solutions to the exact solution is clearly indicated.

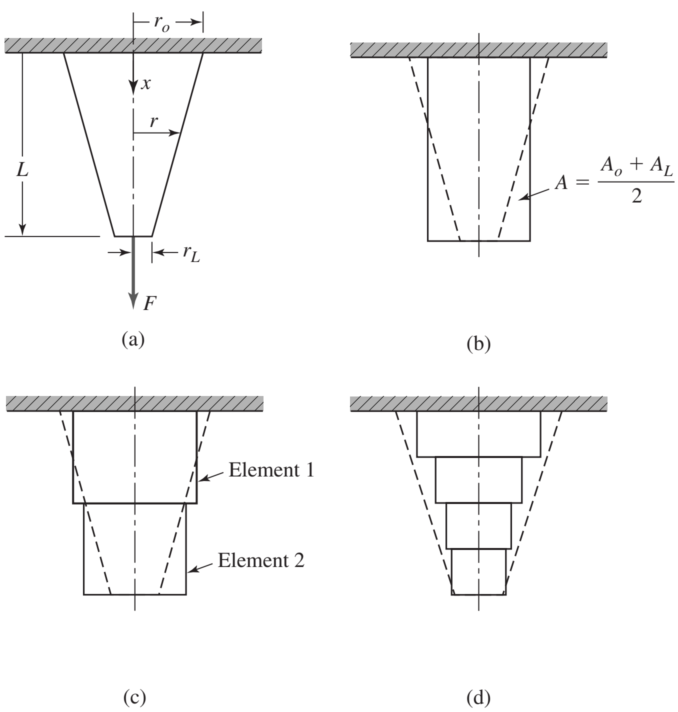


Figure 1.3
(a) Tapered circular cylinder subjected to tensile loading: $r(x) = r_0 - (x/L)(r_0 - r_L)$. (b) Tapered cylinder as a single axial (bar) element using an average area. Actual tapered cylinder is shown as dashed lines. (c) Tapered cylinder modeled as two, equal-length, finite elements. The area of each element is average over the respective tapered cylinder length. (d) Tapered circular cylinder modeled as four, equal-length finite elements. The areas are average over the respective length of cylinder (element length = $L/4$).

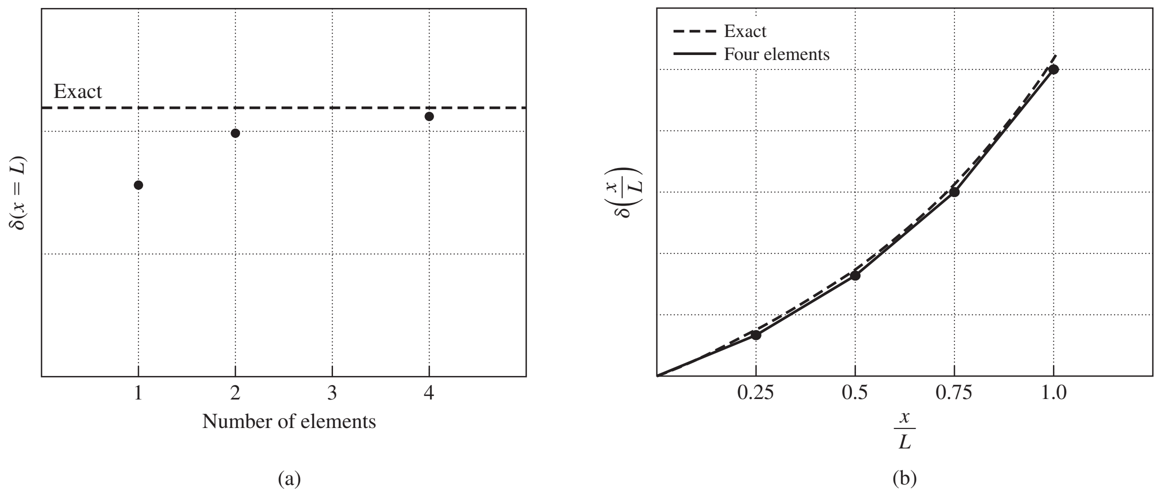


Figure 1.4
(a) Displacement at $x = L$ for tapered cylinder in tension of Figure 1.3. (b) Comparison of the exact solution and the four-element solution for a tapered cylinder in tension.

On the other hand, if we plot displacement as a function of position along the length of the cylinder, we can observe convergence as well as the approximate nature of the finite element solutions. Figure 1.4b depicts the exact strength of materials solution and the displacement solution for the four-element models. We note that the displacement variation in each element is a linear approximation to the true nonlinear solution. The linear variation is directly attributable to the fact that the interpolation functions for a bar element are linear. Second, we note that, as the mesh is refined, the displacement solution converges to the nonlinear solution at *every point* in the solution domain.

The previous paragraph discussed convergence of the displacement of the tapered cylinder. As will be seen in Chapter 2, displacement is the primary field variable in structural problems. In most structural problems, however, we are interested primarily in stresses induced by specified loadings. The stresses must be computed via the appropriate stress-strain relations, and the strain components are derived from the displacement field solution. Hence, strains and stresses are referred to as *derived* variables. For example, if we plot the element stresses for the tapered cylinder example just cited for the exact solution as well as the finite element solutions for two- and four-element models as depicted in Figure 1.5, we observe that the stresses are constant in each element and represent a *discontinuous* solution of the problem in terms of stresses and strains. We also note that, as the number of elements increases, the jump discontinuities in stress decrease in magnitude. This phenomenon is characteristic of the finite element method. The formulation of the finite element method for a given problem is such that the primary field variable is continuous from element to element but

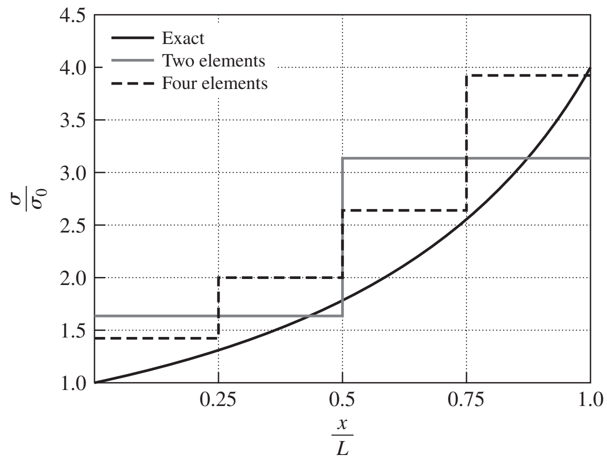


Figure 1.5
Comparison of the computed axial stress value in a tapered cylinder: $\sigma_0 = F/A_0$.

the derived variables are not necessarily continuous. In the limiting process of mesh refinement, the derived variables become closer and closer to continuity.

Our example shows how the finite element solution converges to a *known* exact solution (the exactness of the solution in this case is that of strength of materials theory). If we know the exact solution, we would not be applying the finite element method! So how do we assess the accuracy of a finite element solution for a problem with an unknown solution? The answer to this question is not simple. If we did not have the dashed line in Figure 1.3 representing the exact solution, we could still discern convergence to *a* solution. Convergence of a numerical method (such as the finite element method) is by no means assurance that the convergence is to the correct solution. A person using the finite element analysis technique must examine the solution analytically in terms of (1) numerical convergence, (2) reasonableness (does the result make sense?), (3) whether the physical laws of the problem are satisfied (is the structure in equilibrium? Does the heat output balance with the heat input?), and (4) whether the discontinuities in value of derived variables across element boundaries are reasonable. Many such questions must be posed and examined prior to accepting the results of a finite element analysis as representative of a correct solution useful for design purposes.

1.2.2 Comparison of Finite Element and Finite Difference Methods

The *finite difference* method is another numerical technique frequently used to obtain approximate solutions of problems governed by differential equations. Details of the technique are discussed in Chapter 7 in the context of transient heat transfer. The method is also illustrated in Chapter 10 for transient dynamic analysis of structures. Here, we present the basic concepts of the finite difference method for purposes of comparison.

The finite difference method is based on the definition of the derivative of a function $f(x)$ that is

$$\frac{\mathrm{d}f(x)}{\mathrm{d}x} = \lim_{\Delta x \to 0} \frac{f(x+\Delta x) - f(x)}{\Delta x} \tag{1.2}$$

where x is the independent variable. In the finite difference method, as implied by its name, derivatives are calculated via Equation 1.2 using small, but finite, values of Δx to obtain

$$\frac{\mathrm{d}f(x)}{\mathrm{d}x} \approx \frac{f(x+\Delta x) - f(x)}{\Delta x} \tag{1.3}$$

A differential equation such as

$$\frac{\mathrm{d}f}{\mathrm{d}x} + x = 0 \qquad 0 \le x \le 1 \tag{1.4}$$

is expressed as

$$\frac{f(x+\Delta x) - f(x)}{\Delta x} + x = 0 \tag{1.5}$$

in the finite difference method. Equation 1.5 can be rewritten as

$$f(x+\Delta x) = f(x) - x(\Delta x) \tag{1.6}$$

where we note that the equality must be taken as "approximately equals." From differential equation theory, we know that the solution of a first-order differential equation contains one constant of integration. The constant of integration must be determined such that one given condition (a boundary condition or initial condition) is satisfied. In the current example, we assume that the specified condition is $x(0) = A =$ constant. If we choose an *integration step* Δx to be a small, constant value (the integration step is not *required* to be constant), then we can write

$$x_{i+1} = x_i + \Delta x \qquad i = 0, N \tag{1.7}$$

where N is the total number of steps required to cover the domain. Equation 1.6 is then

$$f_{i+1} = f_i - x_i(\Delta x) \qquad f_0 = A \qquad i = 0, N \tag{1.8}$$

Equation 1.8 is known as a *recurrence relation* and provides an approximation to the value of the unknown function $f(x)$ at a number of discrete points in the domain of the problem.

To illustrate, Figure 1.6a shows the exact solution $f(x) = 1 - x^2/2$ and a finite difference solution obtained with $\Delta x = 0.1$. The finite difference solution is shown at the discrete points of function evaluation only. The manner of variation

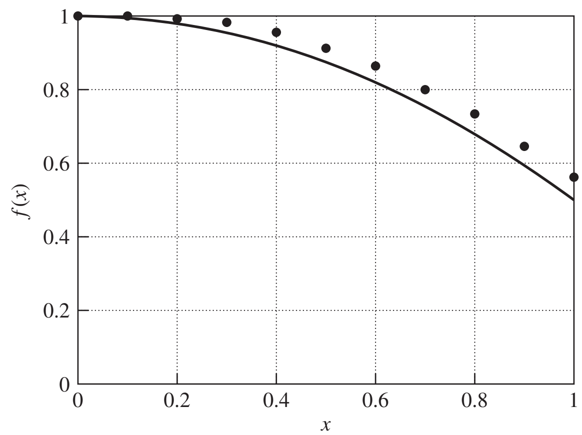


Figure 1.6
Comparison of the exact and finite difference solutions of Equation 1.4 with $f_0 = A = 1$.

of the function between the calculated points is not known in the finite difference method. One can, of course, linearly interpolate the values to produce an approximation to the curve of the exact solution but the manner of interpolation is not an a priori determination in the finite difference method.

To contrast the finite difference method with the finite element method, we note that, in the finite element method, the variation of the field variable in the physical domain is an integral part of the procedure. That is, based on the selected interpolation functions, the variation of the field variable throughout a finite element is specified as an integral part of the problem formulation. In the finite difference method, this is not the case: The field variable is computed at specified points only. The major ramification of this contrast is that derivatives (to a certain level) can be computed in the finite element approach, whereas the finite difference method provides data only on the variable itself. In a structural problem, for example, both methods provide displacement solutions, but the finite element solution can be used to directly compute strain components (first derivatives). To obtain strain data in the finite difference method requires additional considerations not inherent to the mathematical model.

There are also certain similarities between the two methods. The integration points in the finite difference method are analogous to the nodes in a finite element model. The variable of interest is explicitly evaluated at such points. Also, as the integration step (step size) in the finite difference method is reduced, the solution is expected to converge to the exact solution. This is similar to the expected convergence of a finite element solution as the mesh of elements is refined. In both cases, the refinement represents reduction of the mathematical model from finite to infinitesimal. And in both cases, differential equations are reduced to algebraic equations.

Probably the most descriptive way to contrast the two methods is to note that the finite difference method models the differential equation(s) of the problem and uses numerical integration to obtain the solution at discrete points. The finite element method models the entire domain of the problem and uses known physical principles to develop algebraic equations describing the approximate solutions. Thus, the finite difference method models differential equations while the finite element method can be said to more closely model the physical problem at hand. As will be observed in the remainder of this text, there are cases in which a combination of finite element and finite difference methods is very useful and efficient in obtaining solutions to engineering problems, particularly where dynamic (transient) effects are important.

1.3 A GENERAL PROCEDURE FOR FINITE ELEMENT ANALYSIS

Certain steps in formulating a finite element analysis of a physical problem are common to all such analyses, whether structural, heat transfer, fluid flow, or some other problem. These steps are embodied in commercial finite element software packages (some are mentioned in the following paragraphs) and are implicitly incorporated in this text, although we do not necessarily refer to the steps explicitly in the following chapters. The steps are described as follows.

1.3.1 Preprocessing

The preprocessing step is, quite generally, described as defining the model and includes

Define the geometric domain of the problem.
Define the element type(s) to be used (Chapter 6).
Define the material properties of the elements.
Define the geometric properties of the elements (length, area, and the like).
Define the element connectivities (mesh the model).
Define the physical constraints (boundary conditions).
Define the loadings.

The preprocessing (model definition) step is critical. In no case is there a better example of the computer-related axiom "garbage in, garbage out." A perfectly computed finite element solution is of absolutely no value if it corresponds to the wrong problem.

1.3.2 Solution

During the solution phase, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary field variable(s). The computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow.

As it is not uncommon for a finite element model to be represented by tens of thousands of equations, special solution techniques are used to reduce data storage requirements and computation time. For static, linear problems, a *wave front solver,* based on Gauss elimination (Appendix C), is commonly used. While a complete discussion of the various algorithms is beyond the scope of this text, the interested reader will find a thorough discussion in the Bathe book [1].

1.3.3 Postprocessing

Analysis and evaluation of the solution results is referred to as *postprocessing*. Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution. Examples of operations that can be accomplished include

Sort element stresses in order of magnitude.
Check equilibrium.
Calculate factors of safety.
Plot deformed structural shape.
Animate dynamic model behavior.
Produce color-coded temperature plots.

While solution data can be manipulated many ways in postprocessing, the most important objective is to apply sound engineering judgment in determining whether the solution results are physically reasonable.

1.4 BRIEF HISTORY OF THE FINITE ELEMENT METHOD

The mathematical roots of the finite element method dates back at least a half century. Approximate methods for solving differential equations using trial solutions are even older in origin. Lord Rayleigh [2] and Ritz [3] used trial functions (in our context, interpolation functions) to approximate solutions of differential equations. Galerkin [4] used the same concept for solution. The drawback in the earlier approaches, compared to the modern finite element method, is that the trial functions must apply over the *entire* domain of the problem of concern. While the Galerkin method provides a very strong basis for the finite element method (Chapter 5), not until the 1940s, when Courant [5] introduced the concept of piecewise-continuous functions in a subdomain, did the finite element method have its real start.

In the late 1940s, aircraft engineers were dealing with the invention of the jet engine and the needs for more sophisticated analysis of airframe structures to withstand larger loads associated with higher speeds. These engineers, without the benefit of modern computers, developed matrix methods of force analysis, collectively known as the *flexibility method,* in which the unknowns are the forces and the knowns are displacements. The finite element method, in its most often-used form, corresponds to the *displacement method,* in which the unknowns are system displacements in response to applied force systems. In this text, we adhere exclusively to the displacement method. As will be seen as we proceed, the term *displacement* is quite general in the finite element method and can represent physical displacement, temperature, or fluid velocity, for example. The term *finite element* was first used by Clough [6] in 1960 in the context of plane stress analysis and has been in common usage since that time.

During the decades of the 1960s and 1970s, the finite element method was extended to applications in plate bending, shell bending, pressure vessels, and general three-dimensional problems in elastic structural analysis [7–11] as well as to fluid flow and heat transfer [12, 13]. Further extension of the method to large deflections and dynamic analysis also occurred during this time period [14, 15]. An excellent history of the finite element method and detailed bibliography is given by Noor [16].

The finite element method is computationally intensive, owing to the required operations on very large matrices. In the early years, applications were performed using mainframe computers, which, at the time, were considered to be very powerful, high-speed tools for use in engineering analysis. During the 1960s, the finite element software code NASTRAN [17] was developed in conjunction with the space exploration program of the United States. NASTRAN was the first major finite element software code. It was, and still is, capable of hundreds of thousands of degrees of freedom (nodal field variable computations). In the years since the development of NASTRAN, many commercial software packages have been introduced for finite element analysis. Among these are ANSYS [18], ALGOR [19], and COSMOS/M [20]. In today's computational environment, most of these packages can be used on desktop computers and engineering workstations to obtain solutions to large problems in static and dynamic structural analysis, heat transfer, fluid flow, electromagnetics, and seismic response. In this text, we do not utilize or champion a particular code. Rather, we develop the fundamentals for understanding of finite element analysis to enable the reader to use such software packages with an educated understanding.

1.5 EXAMPLES OF FINITE ELEMENT ANALYSIS

We now present, briefly, a few examples of the types of problems that can be analyzed via the finite element method. Figure 1.7 depicts a rectangular region with a central hole. The area has been "meshed" with a finite element grid of two-dimensional elements assumed to have a constant thickness in the z direction. Note that the mesh of elements is irregular: The element shapes (triangles and quadrilaterals) and sizes vary. In particular, note that around the geometric discontinuity of the hole, the elements are of smaller size. This represents not only

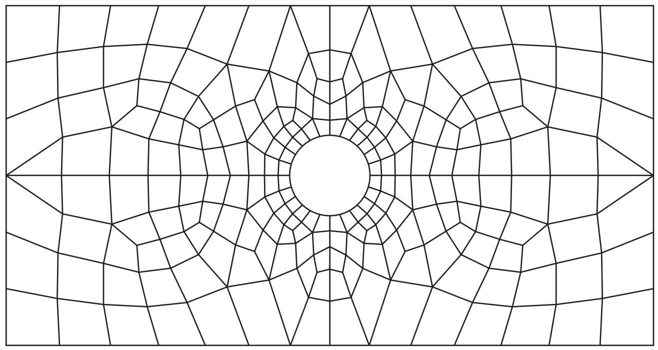


Figure 1.7
A mesh of finite elements over a rectangular region having a central hole.

an improvement in geometric accuracy in the vicinity of the discontinuity but also solution accuracy, as is discussed in subsequent chapters.

The geometry depicted in Figure 1.7 could represent the finite element model of several physical problems. For plane stress analysis, the geometry would represent a thin plate with a central hole subjected to edge loading in the plane depicted. In this case, the finite element solution would be used to examine stress concentration effects in the vicinity of the hole. The element mesh shown could also represent the case of fluid flow around a circular cylinder. In yet another application, the model shown could depict a heat transfer fin attached to a pipe (the hole) from which heat is transferred to the fin for dissipation to the surroundings. In each case, the formulation of the equations governing physical behavior of the elements in response to external influences is quite different.

Figure 1.8a shows a truss module that was at one time considered a building-block element for space station construction [21]. Designed to fold in accordion fashion into a small volume for transport into orbit, the module, when deployed, extends to overall dimensions 1.4 m × 1.4 m × 2.8 m. By attaching such modules end-to-end, a truss of essentially any length could be obtained. The structure was analyzed via the finite element method to determine the vibration characteristics as the number of modules, thus overall length, was varied. As the connections between the various structural members are pin or ball-and-socket joints, a simple axial tension-compression element (Chapter 2) was used in the model. The finite element model of one module was composed of 33 elements. A sample vibration shape of a five-module truss is shown in Figure 1.8b.

The truss example just described involves a rather large structure modeled by a small number of relatively large finite elements. In contrast, Figure 1.9 shows the finite element model of a very thin tube designed for use in heat

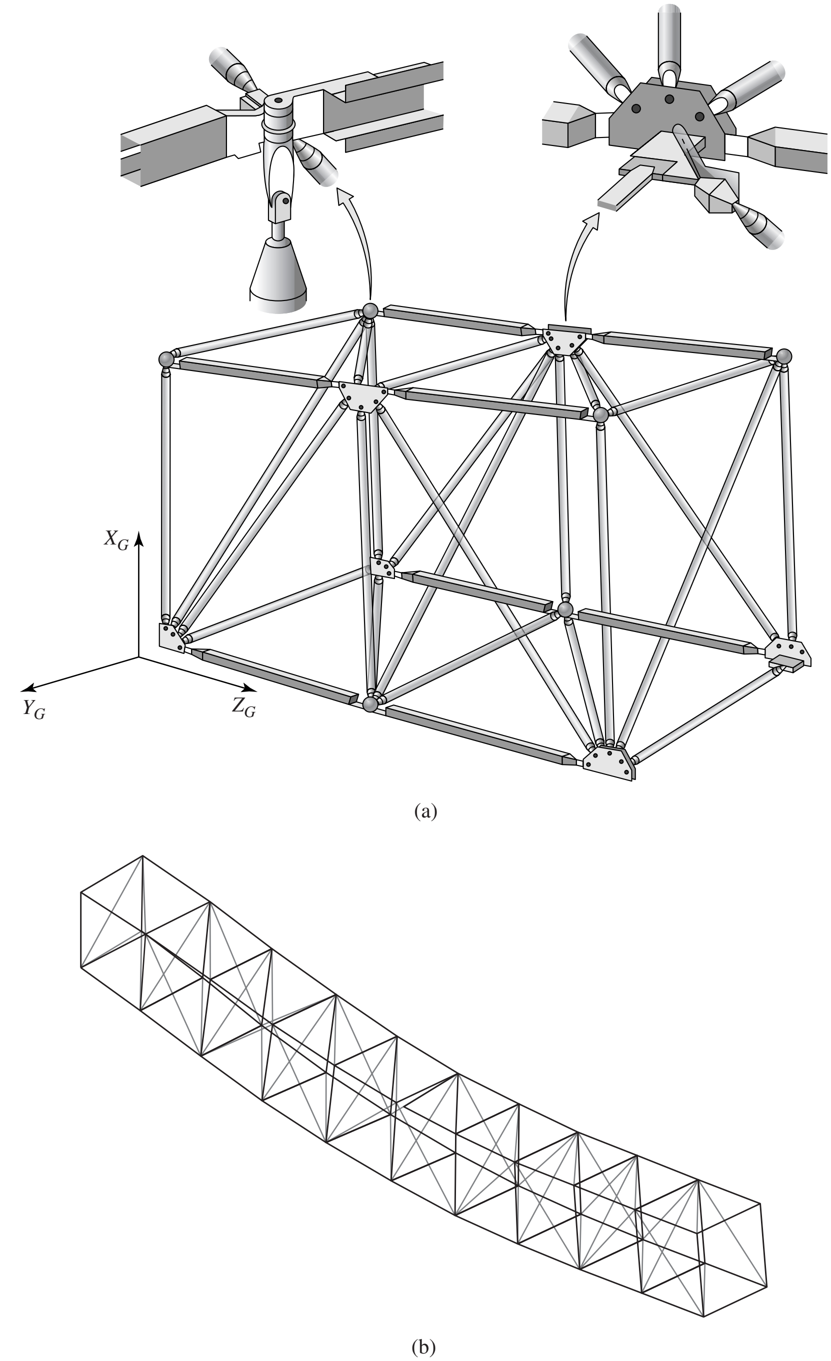


Figure 1.8
(a) Deployable truss module showing details of folding joints.
(b) A sample vibration-mode shape of a five-module truss as obtained via finite element analysis. *(Courtesy: AIAA)*

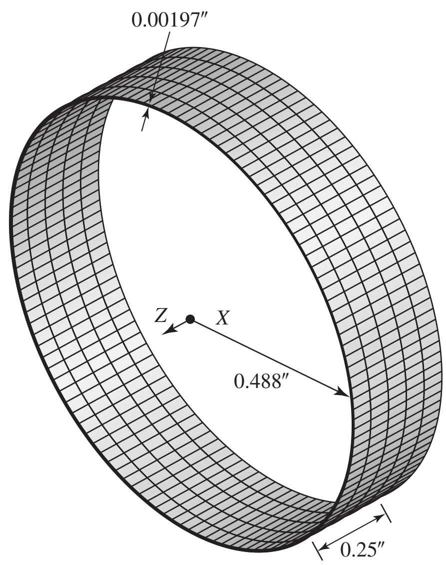
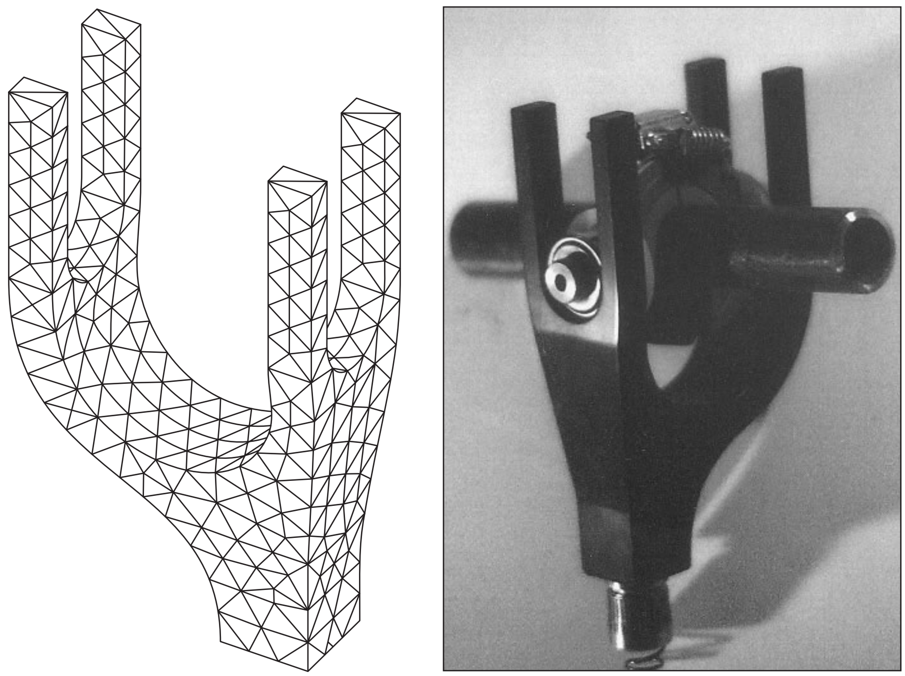


Figure 1.9
Finite element model of a thin-walled heat exchanger tube.

transfer in a spacecraft application. The tube has inside diameter of 0.976 in. and wall thickness 0.00197 in. and overall length 36 in. Materials considered for construction of the tube were copper and titanium alloys. Owing to the wall thickness, prototype tubes were found to be very fragile and difficult to handle without damage. The objectives of the finite element analysis were to examine the bending, torsional, and buckling loads allowable. The figure shows the finite element mesh used to model a section of the tube only 0.25 in. in length. This model contains 1920 three-dimensional solid elements, each having eight nodes with 3 degrees of freedom at each node. Such a large number of elements was required for a small structure in consideration of computational accuracy. The concern here was the so-called *aspect ratio* of the elements, as is defined and discussed in subsequent chapters.

As a final example, Figure 1.10a represents the finite element model of the main load-carrying component of a prosthetic device. The device is intended to be a hand attachment to an artificial arm. In use, the hand would allow a lower arm amputee to engage in weight lifting as part of a physical fitness program. The finite element model was used to determine the stress distribution in the component in terms of the range of weight loading anticipated, so as to properly size the component and select the material. Figure 1.10b shows a prototype of the completed hand design.



(a) (b)

Figure 1.10
(a) A finite element model of a prosthetic hand for weightlifting. (b) Completed prototype of a prosthetic hand, attached to a bar.
(Courtesy of Payam Sadat. All rights reserved.)

1.6 OBJECTIVES OF THE TEXT

I wrote *Fundamentals of Finite Element Analysis* for use in senior-level finite element courses in engineering programs. The majority of available textbooks on the finite element method are written for graduate-level courses. These texts are heavy on the theory of finite element analysis and rely on mathematical techniques (notably, *variational calculus*) that are not usually in the repertoire of undergraduate engineering students. Knowledge of advanced mathematical techniques is not required for successful use of this text. The prerequisite study is based on the undergraduate coursework common to most engineering programs: linear algebra, calculus through differential equations, and the usual series of statics, dynamics, and mechanics of materials. Although not required, prior study of fluid mechanics and heat transfer is helpful. Given this assumed background, the finite element method is developed on the basis of physical laws (equilibrium, conservation of mass, and the like), the principle of minimum potential energy (Chapter 2), and Galerkin's finite element method (introduced and developed in Chapter 5).

As the reader progresses through the text, he or she will discern that we cover a significant amount of finite element theory in addition to application examples. Given the availability of many powerful and sophisticated finite element software packages, why study the theory? The finite element method is a tool, and like any other tool, using it without proper instruction can be quite dangerous. My premise is that the proper instruction in this context includes understanding the basic theory underlying formulation of finite element models of physical problems. As stated previously, critical analysis of the results of a finite element model computation is essential, since those results may eventually become the basis for design. Knowledge of the theory is necessary for both proper modeling and evaluation of computational results.

REFERENCES

1. Bathe, K-J. *Finite Element Procedures*. Englewood Cliffs, NJ: Prentice-Hall, 1996.

2. Lord Rayleigh. "On the Theory of Resonance." *Transactions of the Royal Society (London)* A161 (1870).

3. Ritz, W. "Uber eine neue Methode zur Losung gewissen Variations-Probleme der mathematischen Physik." *J. Reine Angew. Math.* 135 (1909).

4. Galerkin, B. G. "Series Solution of Some Problems of Elastic Equilibrium of Rods and Plates" [in Russian]. *Vestn. Inzh. Tekh.* 19 (1915).

5. Courant, R. "Variational Methods for the Solution of Problems of Equilibrium and Vibrations." *Bulletin of the American Mathematical Society* 49 (1943).

6. Clough, R. W. "The Finite Element Method in Plane Stress Analysis." *Proceedings, American Society of Civil Engineers, Second Conference on Electronic Computation,* Pittsburgh, 1960.

7. Melosh, R. J. "A Stiffness Method for the Analysis of Thin Plates in Bending." *Journal of Aerospace Sciences* 28, no. 1 (1961).

8. Grafton, P. E., and D. R. Strome. "Analysis of Axisymmetric Shells by the Direct Stiffness Method." *Journal of the American Institute of Aeronautics and Astronautics* 1, no. 10 (1963).

9. Gallagher, R. H. "Analysis of Plate and Shell Structures." *Proceedings, Symposium on the Application of Finite Element Methods in Civil Engineering,* Vanderbilt University, Nashville, 1969.

10. Wilson, E. L. "Structural Analysis of Axisymmetric Solids." *Journal of the American Institute of Aeronautics and Astronautics* 3, (1965).

11. Melosh, R. J. "Structural Analysis of Solids." *Journal of the Structural Division,* Proceedings of the American Society of Civil Engineers, August 1963.

12. Martin, H. C. "Finite Element Analysis of Fluid Flows." *Proceedings of the Second Conference on Matrix Methods in Structural Mechanics,* Wright-Patterson Air Force Base, Kilborn, Ohio, October 1968.

13. Wilson, E. L., and R. E. Nickell. "Application of the Finite Element Method to Heat Conduction Analysis." *Nuclear Engineering Design* 4 (1966).

14. Turner, M. J., E. H. Dill, H. C. Martin, and R. J. Melosh. "Large Deflections of Structures Subjected to Heating and External Loads." *Journal of Aeronautical Sciences* 27 (1960).

15. Archer, J. S. "Consistent Mass Matrix Formulations for Structural Analysis Using Finite Element Techniques." *Journal of the American Institute of Aeronautics and Astronautics* 3, no. 10 (1965).

16. Noor, A. K. "Bibliography of Books and Monographs on Finite Element Technology." *Applied Mechanics Reviews* 44, no. 6 (1991).

17. MSC/NASTRAN. Lowell, MA: MacNeal-Schwindler Corp.

18. ANSYS. Houston, PA: Swanson Analysis Systems Inc.

19. ALGOR. Pittsburgh: Algor Interactive Systems.

20. COSMOS/M. Los Angeles: Structural Research and Analysis Corp.

21. Hutton, D. V. "Modal Analysis of a Deployable Truss Using the Finite Element Method." *Journal of Spacecraft and Rockets* 21, no. 5 (1984).