International Journal For Technological Research In Engineering
Volume 1, Issue 12, August-2014 *ISSN (Online): 2347 - 4718*

A CASE STUDY OF THE FINITE ELEMENT METHOD

Dr. P.K.Sharma[1], Prof. Sunil Shukla[2], Deepak Bisen[3]
[1]HOD & Co-Guide, [2]Prof. & Guide, [3]M Tech Scholar, Mechanical Department, NIIST, Bhopal

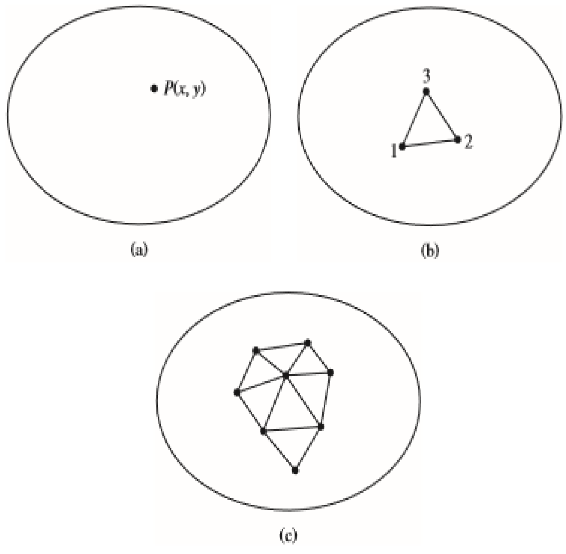
Abstract: Today the finite element method (FEM) is considered as one of the well-established and convenient technique for the computer solution of complex problems in different fields of engineering: civil engineering, mechanical engineering, nuclear engineering, biomedical engineering, hydrodynamics, heat conduction, geo-mechanics, etc. From other side, FEM can be examined as a powerful tool for the approximate solution of differential equations describing different physical processes. The success of FEM is based largely on the basic finite element procedures used: the formulation of the problem in variational form, the finite element dicretization of this formulation and the effective solution of the resulting finite element equations. These basic steps are the same whichever problem is considered and together with the use of the digital computer present a quite natural approach to engineering analysis. The objective of this course is to present briefly each of the above aspects of the finite element analysis and thus to provide a basis for the understanding of the complete solution process. According to three basic areas in which knowledge is required, the course is divided into three parts. The first part of the course comprises the formulation of FEM and the numerical procedures used to evaluate the element matrices and the matrices of the complete element assemblage. In the second part, methods for the efficient solution of the finite element equilibrium equations in static and dynamic analyses will be discussed. In the third part of the course, some modelling aspects and general features of some Finite Element Programs (ANSYS, NISA, LS-DYNA) will be briefly examined.

I. 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.

II. 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 (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 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 computations 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) (b) (c)

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.

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 × 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 1, 2, and 3 are the values of the field variable at the nodes, and N1, N2, and N3 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. If the domain of Figure 1.1 represents a thin, solid body subjected to plane stress, 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.

III. 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, 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.

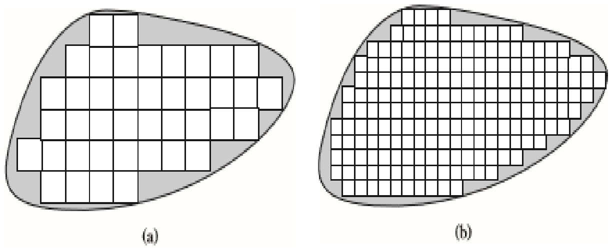

(a) (b)

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.

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 of the cylinder (Figure 1.3b). The uniform bar is a link or bar finite element, 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.

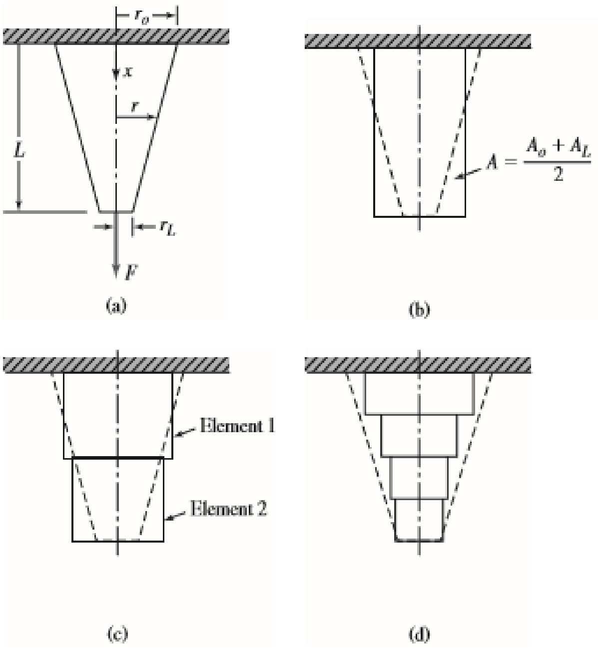


Figure 1.3
(a) Tapered circular cylinder subjected to tensile loading: $r(x) = r_0 - (x/L)(r_0 - r_L)$. (b) Tapered cylinder as a single axial (bar) element using an average area. Actual tapered cylinder is shown as dashed lines. (c) Tapered cylinder modeled as two, equal-length, finite elements. The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

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, 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 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).

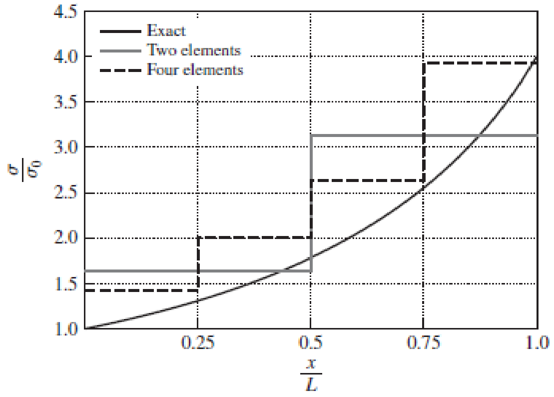


Figure 1.5
Comparison of the computed axial stress value in a tapered cylinder: $\sigma_0 = F/A_0$.

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, (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.

IV. CONCLUSION

FEM was treated previously as a generalization of the displacement method for shaft systems. For a computation of beams, plates, shells, etc. by FEM, a construction is presented in a view of element assembly. It is assumed that they are connected in a finite number of nodal points. Then it is considered that the nodal displacements determine the field of displacements of each finite element. That gives the possibility to use the principle of virtual displacements to write the equilibrium equations of element assembly so, as made for a calculation of shaft systems.

REFERENCE

[1] G.R. Liu, S.S. Quek "Chapter 3 - Fundamentals for Finit Element Method" The Finite Element Method (Second Edition), 2014, Pages 43-79
[2] Zhuo Zhuang, Zhanli Liu, Binbin Cheng, Jianhui Liao "Chapter 1 - Overview of Extended Finite Element Method" Extended Finite Element Method, 2014, Pages 1-12
[3] Gioacchino Alotta, Giuseppe Failla, Massimiliano Zingales "Finite element method for a nonlocal Timoshenko beam model" Finite Elements in Analysis and Design, Volume 89, 15 October 2014, Pages 77-92
[4] Li Shen, Zhangyi Liu, Jiu Hui Wu "B-spline finite element method based on node moving adaptive refinement strategy" Finite Elements in Analysis and Design, Volume 91, 15 November 2014, Pages 84-94
[5] Peter Hansbo, Mats G. Larson, Sara Zahedi "A cut finite element method for a Stokes interface problem"Applied Numerical Mathematics, Volume 85, November 2014, Pages 90-114
[6] J.R. Pettit, A. Walker, P. Cawley, M.J.S. Lowe "A Stiffness Reduction Method for efficient absorption of waves at boundaries for use in commercial Finite Element codes" Ultrasonics, Volume 54, Issue 7, September 2014, Pages 1868-1879