CHAPTER 4.0 FINITE ELEMENT METHOD

4.1 Introduction

Engineers are interested in evaluating effects such as deformations, stresses, temperature, fluid pressure and fluid velocities caused by forces such as applied loads, pressures, thermal or fluid fluxes. The nature of the distribution of the effects or deformations in a body depends on the characteristics of the force system and of the body itself.

It is assumed that it is difficult to find the distribution of effects by using conventional methods and decide to use finite element method, which is based on the concept of discretization as discussed below [7]. The body is divided into a number of smaller regions as illustrated in Figure 4.1. These regions are called finite elements. A consequence of such a subdivision is that the distribution of displacements is also discretized into corresponding subzones as illustrated in Figure 4.1. The subdivided elements are now easier to examine as compared to the entire body and the distribution of effects over it.

4.2 Process Of Discretization

We use a number of terms to process the scheme of discretization such as subdivision, continuity, compatibility, convergence, upper and lower bounds, stationary potential, minimum residual and error. Some of them are :

a. Subdivision

Space is finite and infinitely divisible. All objects must also have magnitude to exist.

b. Continuity

A continuos quantity is made of divisible elements. For instance, there exist other points between any two points in a line and there exist other moments between two moments in a period of time. Therefore, space and time are continuos and infinitely divisible. and things are consecutive and continuos. These ideas of finiteness, divisibility and continuity allow us to divide continuos objects into smaller components, units or elements.

c. Convergence

The process of successively moving towards the exact or correct solution is termed as convergence.

d. Error

It should be apparent that discretization involves approximation. Consequently, what we obtain is not the exact solution but an approximation to that solution. The amount by which we differ can be termed as error.

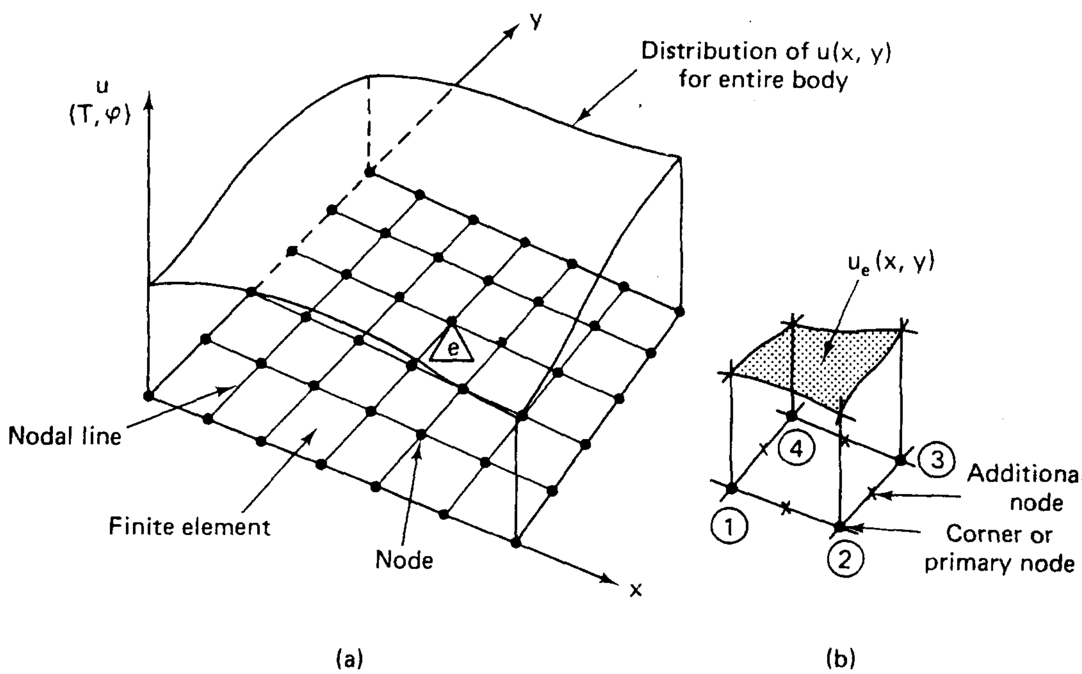

Figure 4.1 : a) Discretization of two dimensional body

b) Distribution of u_e over a generic element e

4.3 Cause And Effect

The essence of all investigations is the examination and understanding of causes and their effects. The effect of work is tiredness and that of too much work is fatigue or stress. The effect of load on a structure is to cause deformations, strains and stresses. Too much load will cause fatigue and failure. In using finite element methods, our main concern is the cause and effects of the forcing function or load on engineering systems.

Finite element method is powerful and popular because it allows solutions of complex problems in engineering and mathematical physics. The complexities arise due to factors such as irregular geometries, nonhomogeneties, non linear behavior and arbitrary loading conditions.

Finite element method is basically divided into 8 steps as discussed below [7] :

Step 1 : Discretize and Select Element Configuration

This step involves subdividing the body into suitable number of small bodies called finite elements. The intersections of the sides of the elements are called nodes or nodal points and the interfaces between the elements are called nodal lines and nodal planes. Refer to Figure 4.1.

An immediate question that arise is number of elements required to approximate the continuos medium a closely as possible.

The type of element to be used will depend on the characteristics of the continuum and the idealization that we may chose to use. For instance, if a structure or body is idealized as one dimensional line as illustrated in Figure 4.2a, then a line element is used. For a two dimensional body, triangles and quadrilaterals as illustrated in Figure 4.2b are used. And for three dimensional idealization, hexahedrons are used as illustrated in Figure 4.2c.

Although the body can be subdivided into regular shaped elements in the interior, special provisions may be required if the boundary is irregular. For many cases, the irregular boundary can be approximated by a number of straight lines.

On the other hand, for many problems it may be necessary to use mathematical functions of sufficient orders to approximate the boundaries. For example, if the boundary shape is similar to a parabolic curve, a second order quadratic function can be used.

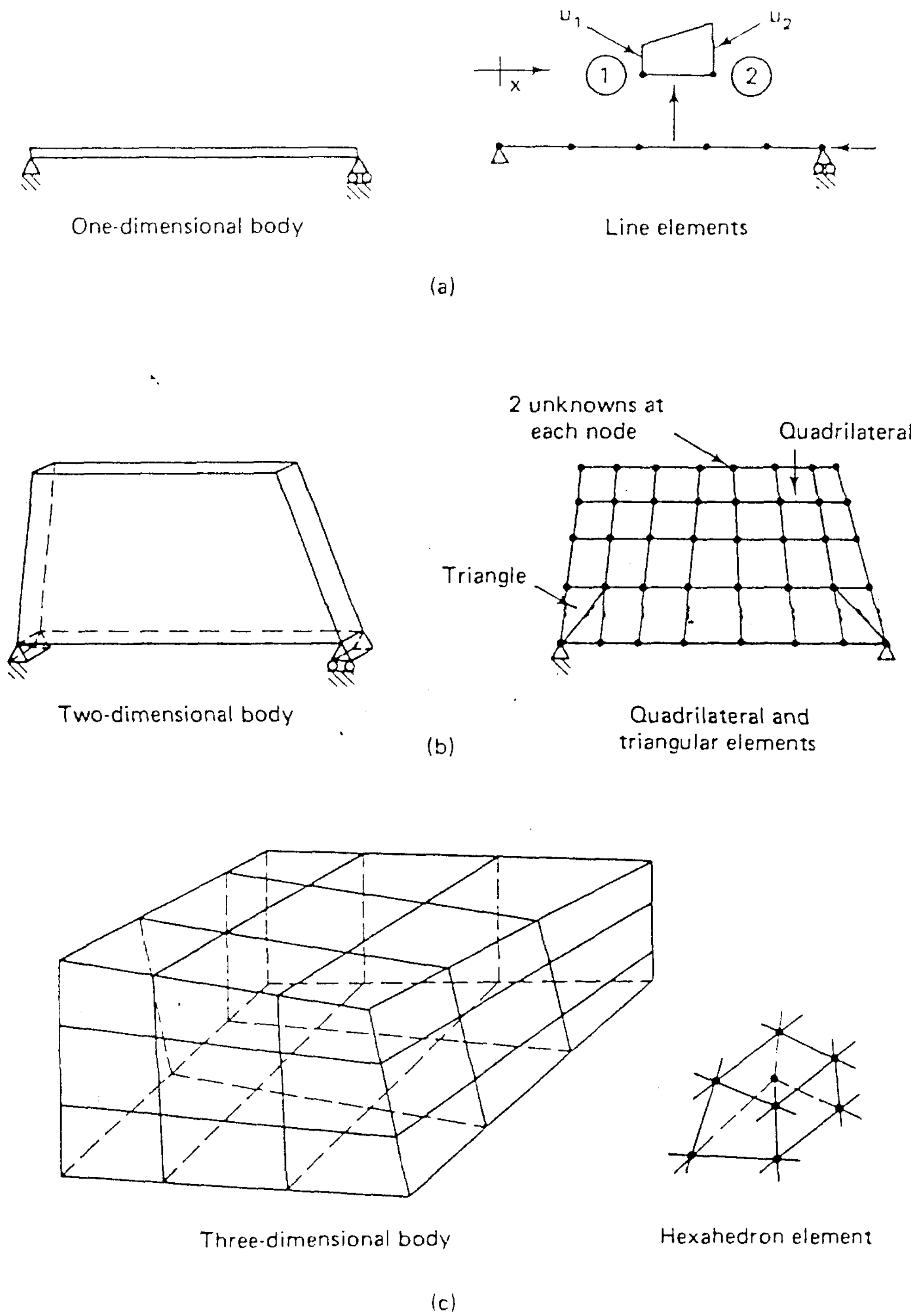

Figure 4.2 : Different types of elements

a) One dimensional elements b) Two dimensional elements

c) Three dimensional elements

Step 2 : Select Approximation Models or Functions.

In this step, a pattern or shape for the distribution of the unknown quantity is chosen. It can be that of a displacement and or stress deformation problems, temperature in heat flows, fluid pressures and or velocity for fluid flow problems and both temperature and displacement for coupled problems involving effects of both flow and deformation [7] .

The nodal points of the element provide strategic points for writing mathematical functions to describe the shape of the distribution of the unknown quantity over the domain of the element. A number of mathematical functions such as polynomial and trigonometric series can be used for this purpose especially polynomials because of the ease and simplifications they provide in the finite element formulation If we denote u as ıe unknown, the polynomial interpolation function can be expressed as :

$$u = N_1 u_1 + N_2 u_2 + N_3 u_3 + \ldots\ldots\ldots N_m u_m \qquad \textbf{(4.1)}$$

Here u_1, u_2, u_3 ....u_m are the values of the unknowns at the nodal points and N_1, N_2, N_3..........N_m are the interpolation functions [7] .

After all the steps of the finite element methods are accomplished, the values of the unknowns u at all the nodes that is u_1, u_2, u_3 ....u_m are determined. To initiate action towards obtaining the solution, however, an assumption is made in advance of a shape or pattern that will satisfy the conditions, laws and principles of the problem at hand.

The solution obtained will be in the terms of the unknowns only at the nodal points. this is one of the outcomes of the discretization process. Figure 4.3 shows that the final solution is a combination of solutions in each element patched together at the common boundaries. This is further illustrated by sketching a cross section along A-A. It can be seen that the computed solution is not necessarily the same as the exact continuos solution shown by the solid curve. This is due to the fact that discretization only yields approximate solutions. However, discretization can be manipulated to ensure that the computed solution is as close to the exact solution.

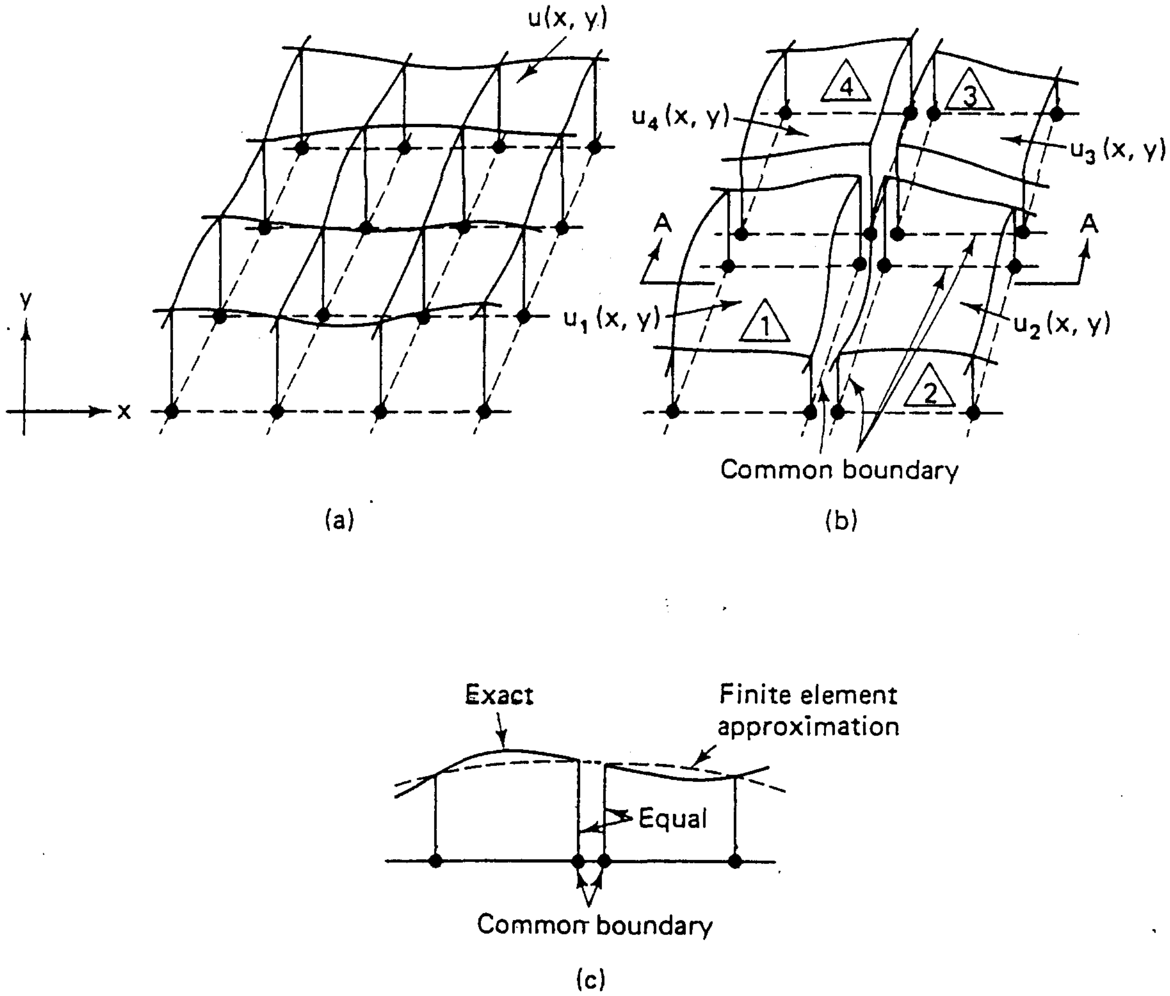

Figure 4.3 : Approximate solution as patchwork of solutions

a) assemblage b) neighbouring elements c) section A-A

Step 3 : Define Strain -Displacement and Stress-Strain Relationships

In finite element method, appropriate quantities that are used in the governing principles must be defined. For stress deformation problems [7] , one such quantity is the strain or gradient of displacement. For instance, in the case of deformation occurring only in one direction y as illustrated in Figure 4.4, the strain e_y, assumed to be small, is given by

$$e_y = dv / dy \qquad (4.2)$$

where v is the deformation in the y direction [7].

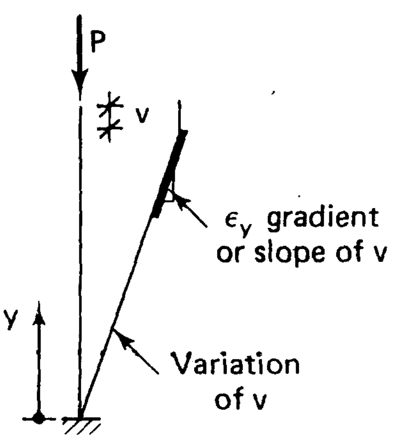

Figure 4.4 : One dimensional stress deformation

In addition to strain or gradient, the stress must also be defined.. This is usually done by expressing its relationship with the strain. Such a relation is called a stress-strain law. In a generalized sense, it is a constitutive law and describes the response or effect(displacement, strain) in a system due to applied cause or force. The stress-strain law is one of the most vital parts of the finite element analysis. Unless it is defined to reflect precisely the behavior of the material or the system, the results from the analysis can of little significance. As an elementary illustration, consider Hooke's law which defines the relationship of stress to strain in a solid body [7] :

$$\sigma_y = E_y e_y \qquad \textbf{(4.3)}$$

where σ_y = stress in the vertical direction

E_y = Young's Modulus of Elasticity.

If e_y from equation 4.2 is subsituted into equation 4.3, the expression of stress in terms of displacements is obtained as shown below [7] ;

$$\sigma_y = E_y \times (dv / dy) \qquad \textbf{(4.4)}$$

Step 4 : Derive Element Equations

By invoking available laws and principles, equations governing the behavior of the element are obtained. The equations are obtained in general terms and hence can be used for all elements in the discretized body.

A number of alternatives are possible for the derivation of element equations. The two most commonly used are the energy methods and the residual methods.

Use of either of the methods will lead to equations describing the behavior of an element, which are commonly expressed as shown below [7] :

$$[\,k\,]\,\{\,q\,\} = \{Q\} \tag{4.5}$$

where $[\,k\,]$ = element property matrix

$\{\,q\,\}$ = vector of unknowns at the element nodes

$\{\,Q\,\}$ = vector of element nodal forcing function

Step 5 : Assemble Element Equations To Obtain Global or Assemblage Equations And Introduce Boundary Conditions.

The final objective is to obtain equations for the entire body that define approximately the behavior of the entire body or structure.

Once the element equations are established for a generic element, equations for other elements are generated recursively by using the same equations again and again. Then all are added together to find the global equations. This assembling process is based on the law of compatibility or continuity. It requires that the body remain continuos, that is, the neighbouring points should remain at the neighbourhood of each other after the load is applied. In other words, the displacements of the two adjacent or consecutive points must have identical values. Depending on the type and nature of the problem, there is a need to enforce the continuity conditions more seriously.

For instance for deformations occurring in a plane, it may be sufficient to enforce continuity of the displacements only. On the other hand, for bending problems, the physical properties of the deformed body under the load requires that in addition to the continuity of the displacements, the slopes or the first derivative of the displacements must also be continuos or compatible at adjacent nodes . Often it may be necessary to satisfy compatibility of the curvatures or the second derivative also.

Finally, the assemblage equations are obtained. These equations are expressed in matrix notation as stated below [7] :

$$[K]\{r\} = \{R\} \tag{4.6}$$

where $[K]$ = assemblage property matrix

$\{r\}$ = assemblage vector of nodal unknowns

$\{R\}$ = assemblage vector of nodal forcing parameters.

Equation 4.6 tells us the capability of the body to withstand the applied forces. In the case of engineering bodies, the surroundings or the constraints of the body are known as the boundary conditions. Only when these conditions are introduced, a decision can be made on how the body will perform.

Boundary conditions are the physical constraints or supports that must exist so that the structure or body can stand in place uniquely. These conditions are commonly specified in terms of known values of the unknowns on a part of the surface or boundary and/or derivatives of the unknowns.

To reflect the boundary conditions in the finite element approximation of the body represented by Eq 4.6, it is usually necessary to modify the equations for the geometric boundary conditions. The final modified assemblage equations are expressed by inserting overbars as shown below [7] :

$$[\bar{K}]\{\bar{r}\} = \{\bar{R}\} \tag{4.7}$$

Step 6 : Solve For The Primary Unknowns

Equation 4.7 is a set of linear or nonlinear simultaneous algebraic equations which can be written in a standard familiar form as :

$$K_{11}r_1 + K_{21}r_2 + \ldots\ldots\ldots\ldots\ldots\ldots K_{1n}r_n = R_1 \qquad \textbf{(4.8)}$$

$$K_{21}r_1 + K_{22}r_2 + \ldots\ldots\ldots\ldots\ldots\ldots K_{2n}r_n = R_2 \qquad \textbf{(4.9)}$$

$$\vdots$$

$$K_{n1}r_1 + K_{n2}r_2 + \ldots\ldots\ldots\ldots\ldots\ldots K_{nn}r_n = R_n$$

The above equations will be solved for unknowns(displacements)r_1, r_2, r_3 ....r_n. These are called primary unknowns because they appear as the first quantities sought in the basic Eq 4.8. The designation of the word primary will change depending on the unknown quantity that appears in Eq 4.8. For example if the problem is formulated by using stresses as unknowns, then the stresses will be called the primary quantities.

Step 7 : Solve For Derived Or Secondary Quantities

Very often additional or secondary quantities must be computed from the primary quantities. In the case of stress-deformation problems, such quantities can be strains, moments, and shear forces. For the flow problems, they can be velocities and discharges. It is relatively straightforward to find the secondary quantities once the primary quantities are known since we can make use of the relations between the strain and displacement and stress and strain that are defined in Step 3.

Step 8 : Interpretation Of Results

The final and the important aim is to reduce the results from the use of the finite element procedure to a form that can be readily used for analysis and design. The results are used to select critical sections of the body and plot the values of displacements and stresses along them.