SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF BUILDING AND ENVIRONMENT

DEPARTMENT OF CIVIL ENGINEERING

UNIT – I - INTRODUCTION – SCIA5201

UNIT-I INTRODUCTION

Boundary Conditions

For structural mechanics problems the boundary conditions may be kinematic, i.e. where the displacements (and slope, i.e. derivative of displacement), may be prescribed, or static, i.e. where forces (and moments) may be prescribed. In problems where time is involved the initial values may have to be specified. Figure 1.1 shows a cantilever beam AB subjected to a uniformly distributed load. If the vertical deflection w at any point is taken as a field variable, it must satisfy the differential equation Eq. (1.1), which is an equilibrium condition,

$$EI \, d^4 w / dx^4 = p$$

And the solution to the above equation must also satisfy the boundary conditions at A and B as follows.

Introduction

Kinematic boundary conditions at A: Displacement w=0 and slope

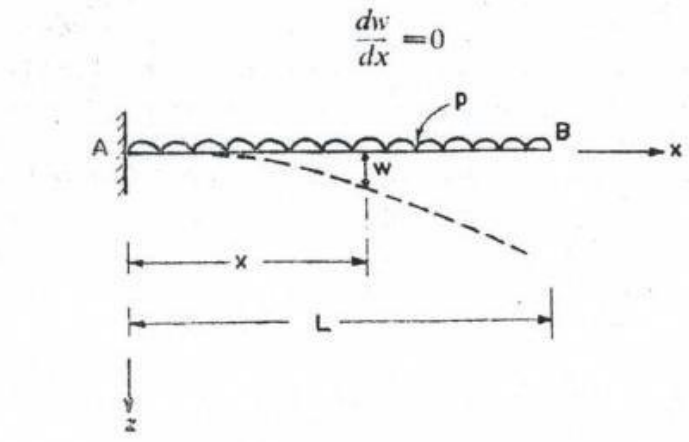


Fig. 1.1 A cantilever beam

static or force boundary conditions at B: shear force,

$$EI \frac{d^3w}{dx^3} = 0$$

and bending moment

$$EI \frac{d^2w}{dx^2} = 0$$

The variational formulation of the above problem is discussed and the application of Euler-Lagrange equation to obtain the governing differential equation is illustrated for the above cantilever beam problem.

APPROXIMATE SOLUTIONS

It is not possible to obtain analytical solution for many engineering problems. An analytical solution is a mathematical expression that gives the value of the field variable at any location in the body. For problems involving complex shapes, material properties and complicated boundary conditions, it is difficult and if many cases intractable to obtain analytical solution that satisfies the governing differential equations or gives extreme value to the governing functional. Hence, for mast of the practical problems the engineer resorts to numerical methods that provide approximate but acceptable solutions. The three methods that are used are as follows:
(i) Functional approximation (ii) Finite difference method (iii) Finite element method
A brief description of the first two methods is given in the subsequent sections and then the finite element method is introduced as a powerful numerical method, widely used in practice.

Functional Approximation

A set of independent functions satisfying the boundary conditions is chosen and a linear combination of a finite number of them is taken to approximately specify the field variable at any point. The unknown parameters that combine the functions are found out in such a way to achieve at best the field condition, which is represented through variational formulation. The well known classical methods such as Rayleigh- Ritz, Galerkin and collocation are based on functional approximation but vary in their procedure for valuating the unknown parameters [6].

The Ray-leigh Ritz method is briefly described below. Consider a simply supported beam, shown in Fig.1.2

subjected to a central concentrated load P and a uniformly distributed load of intensity

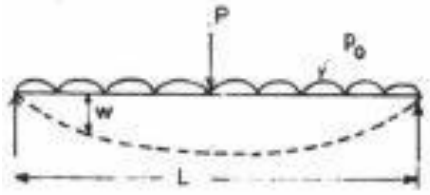


Fig. 1.2 Simply supported beam

In this problem if the deflected shape of the beam is known, the bending moment and shear force at any cross-section can be determined. Consider the following approximation to the deflection curve that satisfies the boundary condition.

$$W=a1\ \mathrm{Sin}\frac{\pi x}{L}+a2\frac{3\pi x}{L} \qquad (a)$$

where a1 and a2 are unknown parameters. It is known from the elementary strength of materials that the strain energy, U, of the beam due to bending is

$$U=\frac{EI}{2}\int_0^L\left(\frac{d^2w}{dx^2}\right)^2 dx \qquad (b)$$

Substituting for w from equation (a) into equation (b),

$$U=\frac{EI}{2}\int_0^L\left(\frac{-\pi^2 a_1}{L^2}\sin\frac{\pi x}{L}-\frac{9\pi^2 a_2}{L^2}\sin\frac{3\pi x}{L}\right)^2 dx \qquad (c)$$

$$=\frac{EI\pi^4}{4L^3}(a_1^2+81\,a_2^2)$$

The potential energy due to loads is given by,

$$H = -\int_0^L P0\, w\, dx - Pwmax$$

$$= -\int_0^L p_0 \left(a_1 \sin \frac{\pi x}{L} + a_2 \sin \frac{3\pi x}{L} \right) dx - P(a_1 - a_2)$$

$$= \frac{-2p_0 L}{\pi} \left(a_1 + \frac{a_2}{3} \right) - P(a_1 - a_2) \qquad \text{(d)}$$

The total potential energy, π, of the beam is

$$\pi = U + H$$

$$= \frac{EI\pi^4}{4L^3} (a_1^2 + 81\, a_2^2) - \frac{2p_0 L}{\pi} \left(a_1 + \frac{a_2}{3} \right) - P(a_1 - a_2) \qquad \text{(e)}$$

For stable equilibrium of the body the potential energy attains stationary value. It can be seen from equation (e) that the total potential energy is now expressed in terms of the parameters al and a2. Hence, for stationary value of 7T the following conditions must be satisfied,

$$\frac{\partial \pi}{\partial a1} = 0 \qquad \frac{\partial \pi}{\partial a2} = 0 \tag{f}$$

Applying the above conditions to equation (e) we get,

(g)

$$\frac{EI\,\pi^4}{2\,L^3}\,a_1 - \frac{2\,p_0\,L}{\pi} - P = 0 \tag{h}$$

$$\frac{81\,EI\,\pi^4}{2\,L^3}\,a^2 - \frac{2\,p_0\,L}{3\,\pi} + P = 0$$

Solving the above two equations we get, (i)

$$a_1 = \frac{2\,L^3}{EI\,\pi^4}\left(\frac{2\,p_0\,L}{\pi} + P\right)$$

$$a_2 = \frac{2\,L^3}{81\,EI\,\pi^4}\left(\frac{2\,p_0\,L}{3\pi} - P\right) \tag{j}$$

Thus the maximum deflection at the centre of the beam is,

$$w_{max} = a_1 - a_2$$

$$w_{max} = \frac{P\,L^3}{48.11\,EI} + \frac{p_0\,L^4}{76.82\,EI} \tag{k}$$

The maximum deflection practically coincides with the exact value of

$$\frac{P\,L^3}{48\,EI} + \frac{p_0\,L^4}{76.8\,EI}$$

The bending moment at any point along the beam is given by (l)

$$M_x = EI\frac{d^2 w}{dx^2}$$

After substituting for the values of at and a2 from equation (i) and (j) into equation (a) and differentiating, the bending moment, Me, at the centre of the beam, x = L/2, can be shown to be

$$M_c = \frac{PL}{4.44} + \frac{p_0 L^2}{8.05} \qquad \text{(m)}$$

It may be noted that the error in the first term is 9.92 per cent and that in the second term is 0.62 per cent. This error can be reduced by adding more terms to the approximate (or trial) function for w, i.e. into equation (a). The above procedure can be

$$u = a_1 \phi_1 (x, y, z) + a_2 \phi_2 (x, y, z) + \ldots + a_n \phi_n (x, y, z)$$
$$v = b_1 \beta_1 (x, y, z) + b_2 \beta_2 (x, y, z) + \ldots + b_n \beta_n (x, y, z)$$
$$w = c_1 \psi_1 (x, y, z) + c_2 \psi_2 (x, y, z) + \ldots + c_n \psi_n (x, y, z)$$

extended to the analysis of a three dimensional solid. In general a deformable body consists of infinite material points and, therefore, it has infinitely many degrees of freedom. By the Rayleigh-Ritz method such a continuous system is reduced to a system of finite degrees of freedom. For the case of three dimensional solid, the variation of the field variables, displacements u, r and w can approximately be represented by the following trial functions

where ai, th and ci are linearly independent unknown parameters and 0, p (x, y, z), , (x,y,z) and 1bi (x, v, z), where 2, . . n are continuous functions in x, y and z that satisfy all the kinematic boundary conditions. By this approximation the body is reduced to have

$$\pi (u, v, w) = \pi (x, y, z, a_1 \ldots a_n, b_1 \ldots b_n, c_1 \ldots c_n)$$

311 degrees of freedom. Now the potential energy of the body can be expressed by a functional in terms of these parameters.

As was indicated earlier that for stable equilibrium of the body the potential energy attains a stationary value and as the potential energy functional is in terms of the parameters al, bi and ci the following equations must be satisfied.

$$\frac{\partial \pi}{\partial a_1} = 0 \qquad \frac{\partial \pi}{\partial a_2} = 0 \ldots \frac{\partial \pi}{\partial a_n} = 0$$
$$\frac{\partial \pi}{\partial b_1} = 0 \qquad \frac{\partial \pi}{\partial b_2} = 0 \ldots \frac{\partial \pi}{\partial b_n} = 0$$
$$\frac{\partial \pi}{\partial c_1} = 0 \qquad \frac{\partial \pi}{\partial c_2} = 0 \ldots \frac{\partial \pi}{\partial c_n} = 0$$

we get 3,1 linear algebraic equations to solve the u., known parameters ai, bt and Ci. It may be noted that the assumed trial functions must be continuous and satisfy all the prescribed boundary conditions, and no simple guidelines are available to select such functions. Also the classical approach of arriving at the equations of the type (1.6) is quite cumbersome. Hence, except in simple situations, this approach could not beused to solve practical problems. However, the concepts used in Rayleigh-Ritz method, i.e. representing the variation of the field variable by trial function andfinding the unknown parameters through minimization of potential energy, are well exploited in the finite element method.

Variational method

Variational formulation is the generalized method of formulating the element stiffness matrix and load vector using the variational principle of solid mechanics. The strain energy in a structural body is given by the relation

$$U = \frac{1}{2}\iiint_{\Omega} \varepsilon^T \sigma \, d\Omega$$

For a 3D structural problem, stress has six components: $\{\sigma\} = \{\sigma_x, \sigma_y, \sigma_z, T_{xy}, T_{yz}, t_{zx}\}$
Similarly, there are six components of strains: $\{\varepsilon\}^T = \{\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{zx}\}$
Now the strain displacement relationship can be expressed as $\{\varepsilon\} = [B]\{d\}$, where $\{d\}$ is the displacement vector in x, y and z directions and [B] is called as the strain displacement relationship matrix. Again, the stress can be represented in terms of its constitutive relationship matrix: $\{\sigma\} = [D]\{\varepsilon\}$. Here [D] is called as the constituent relationship matrix. Using the above relationship in the strain energy equation one can arrive

$$U = \frac{1}{2}\iiint_{\Omega} d^T B^T D B d \, d\Omega$$

Applying the variational principle one can express

$$\{F\} = \frac{\partial u}{\partial d} = \iiint [B]^T [D][B] d\Omega \{d\}$$

Now, from the relationship of $\{F\} = [K]\{d\}$, one can arrive at the element stiffness matrix as:

$$[K] = \iiint_{\Omega} B^T [D][B] \, d\Omega$$

Thus, by the use of Variational principle, the stiffness matrix of a structural element can be obtained as expressed in the above equation.

Weighted Residual Method

Virtual work and Variational method are applicable and adequate for most of the problems. However, in some cases functional analogous to potential energy cannot be written because of not having clear physical meaning. For some applications, such as in fluid mechanics problem, functional needed for a variational approach cannot be expressed. For some types of fluid flow problems, only differential equations and boundary conditions are available. For Such problems weighted residual method can be used for obtaining the solutions. Approximate solutions of differential equation satisfy only part of conditions of the problem. For example a differential equation may be satisfied only at few points, rather than at each. The strategy used in weighted residual method is to first take an approximate solution and then its validity is assessed. The different methods in weighted Residual Method are

- Collocation method
- Least square method
- Method of moment
- Galerkin method

The mathematical statement of a physical problem can be defined as: In domain Ω,

$Du - f = 0$

Where,

D is the differential operator
u = u(x) = dependent variables such as displacement, pressure, velocity, potential function
x = independent variables such as coordinates of a point f = a function of x which may be constant or zero

If u is an approximate solution then residual in domain Ω, $R = Du - f$
According to the weighted residual method, the weak form of above equation will become

$$\int_{\Omega} W_i \, R \, d\Omega = 0 \quad \text{For } i = 1,2,3,....n$$

or

$$\int_{\Omega} W_i \, (Du - f) \, d\Omega = 0$$

Where weighting function $w_i = w_i(x)$ is chosen from the approximate basis function used for constructing approximated solution u.

RAYLEIGH-RITZ METHOD

In mechanics of solids, our problem is to determine the displacement u of the body satisfying the equilibrium stresses are related to strains, which, in turn, are related to displacements. This leads to requiring solution of second order partial differential equations. Solution of this set of equations is generally referred to as an *exact* solution. Such exact solutions are available for simple geometries and loading conditions, and one may refer to publications in theory of elasticity. For problems of complex geometries and general boundary and loading conditions, obtaining such solutions is an almost impossible task. Approximate solution methods usually employ potential energy or variational methods, which place less stringent conditions on the functions.

Potential Energy

The total potential energy of an elastic body, is defined as the sum of total strain energy *(U)* and the work potential:

$$n = \underset{(U)}{\text{Strain energy}} + \underset{(WP)}{\text{Work potential}}$$

For linear elastic materials., the strain energy per unit volume in the body is $\frac{1}{2}\sigma^T\varepsilon$.

The total strain energy *U* is given by

$$U=\frac{1}{2}\int_v \sigma^T \varepsilon \, dv$$

The work potential WP is given by

$$Wp = -\int_v U^T f \, dV - \int_s U^T T ds - \sum_i u \, T_i \, P_j$$

The total potential for the general elastic body is

$$n = \frac{1}{2}\int_v \sigma^T dV - \int_v u^T f dV - \int_s u^T dS - \sum_f u^T{}_i p_i$$

We consider conservative systems here, where the work potential is independent of the path taken. In other words, if the system is displaced from a given configuration and brought back to this state, the forces do zero work regardless of the path.

Kinematically admissible displacements are those that satisfy the single-valued nature of displacements (compatibility) and the boundary conditions. In problems where displacements are the unknowns, which is the approach in this book, compatibility is automatically satisfied.

Galerkins method

Galerkin method is the most widely used among the various weighted residual methods. Galerkin method incorporates differential equations in their weak form, i.e., before starting integration by parts it is in strong form and after by parts it will be in weak form, so that they are satisfied over a domain in an integral. Thus, in case of Galerkin method, the equations are satisfied over a domain in an integral or average sense, rather than at very point. The solution of the equations must satisfy theboundary conditions. There are two types of boundary conditions:

- Essential or kinematic boundary condition
- Non essential or natural boundary condition

As a result, displacement and slope will be essential boundary condition where as moment and shear will be non-essential boundary condition.

Galerkin's method uses the set of governing equations in the development of an integral form. It is usually presented as one of the weighted residual methods. For our discussion, let us consider a general representation of a governing equation on a region *V:*

$$Lu = P \qquad (a)$$

For the one-dimensional rod the governing equation is the differential equation

$$\frac{d}{dx}\left(EA\frac{du}{dx}\right) = 0$$

We may consider *L* as the operator

$$\frac{d}{dx}EA\frac{d}{dx}(\ )$$

operating on *u.*

The exact solution needs to satisfy (a) at every point *x.* If we seek an approximate solution u, it introduces an error ε(x), called the *residual:*

$$\varepsilon(X) = Lu - P \text{ (b)} \qquad (b)$$

The approximate methods revolve around setting the residual relative to a weighting function w_i to zero:

$$\int W_i(LU - P)dV = 0 \qquad i = 1 \text{ to } n \qquad (c)$$

The choice of the weighting function Wi leads to various approximation methods. In the Galerkin method, the weighting functions W_i are chosen from the basis functions used for constructing U. Let u be represented by

$$u=2\sum_{i=1}^{n} Q_iG_i \qquad (d)$$

where Gi *i* = 1 to *n,* are basis functions (usually polynomials of *x,* y, *z).* Here, we choose the weighting functions to be *a linear combination of the basis functions* G_i Specifically, consider an arbitrary function given by

$$\Phi = \sum_{i=1}^{n} \Phi_i G_i \qquad \text{(e)}$$

itrary,

where the coefficients Φ_i are arbitrary, except for requiring that Φ_i satisfy homogeneous (zero) boundary conditions where *u* is prescribed.

Galerkin's method can be stated as follows:
Choose basis functions *Gi* · Determine the coefficients Q_{ii} in u $= \sum_{i=1}^{n} Q_i G_i$ such that

$$\int \Phi(LU - P)\, dV = 0 \qquad \text{(f)}$$

for every Φ of the type $\Phi = \sum_{i}^{n} \Phi_i G_i$ where coefficients Φ, are arbitrary except for requiring that Φ satisfy homogeneous (zero) boundary conditions. The solution of the resulting equations for Qi then yields the approximate solution U.

Usually, in the treatment of (f) an integration by parts is involved. The order of the derivatives is reduced and the natural boundary conditions, such as surface-force conditions, are introduced.

Displacement

In the case of a three dimensional solid or a structure, the displacement at any point can be expressed by its components u, v and w parallel to the cartesian coordinate axes x, y and z. The displacement u, v and w are continuous functions of x, y and z. Any virtual displacement will also be continuous functions and in addition that they should satisfy the kinematic boundary conditions.

The equilibrium conditions to be satisfied by the stresses within the body and on the boundary are given in Eqs:

$$\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + X_b = 0$$

$$\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + Y_b = 0 \qquad \text{(a)}$$

and

$$\sigma_x l + \tau_{xy} m = X_s$$

$$\tau_{xy} l + \sigma_y m = Y_s \qquad \text{(b)}$$

Multiply the equilibrium Eqs 2.5 by Su and 8v respectively and integrate over the area of the solid

$$\int\int \left[\left(\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + X_b \right) \delta u + \left(\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + Y_b \right) \delta v \right] dx dy = 0 \qquad \text{(c)}$$

In order to give a physical interpretation to the equation we can consider Su and 8v as virtual displacements and also expand each one of the terms by using the Green's theorem in two dimensions. if sb(x, y) and 0(x, y) are continuous functions with their first and second partial derivatives are also continuous, then according to Green's theorem,

$$\begin{aligned}&\iint\left(\frac{\partial\phi}{\partial x}\cdot\frac{\partial\psi}{\partial x}+\frac{\partial\phi}{\partial y}\cdot\frac{\partial\psi}{\partial y}\right)dxdy\\&=-\iint\phi\left(\frac{\partial^2\psi}{\partial x^2}+\frac{\partial^2\psi}{\partial y^2}\right)dxdy\\&\quad+\int\phi\left(\frac{\partial\psi}{\partial x}l+\frac{\partial\psi}{\partial y}m\right)dS\end{aligned}\qquad\text{(d)}$$

Consider the first term of the integral in eq (c). Let $\Phi=\sigma x$ and assume ψ such that

$$\frac{\partial\psi}{\partial x}=\delta u \qquad \text{and} \qquad \frac{\partial\psi}{\partial y}=0.$$

Then

$$\iint\frac{\partial\sigma_x}{\partial x}\,\delta u\,dxdy=-\iint\sigma_x\frac{\partial\delta u}{\partial x}\,dxdy+\int\sigma_x l\,\delta u dS$$

Similarly it can be shown that

$$\iint\frac{\partial\sigma_y}{\partial y}\,\delta v\,dxdy=-\iint\sigma_y\frac{\partial\delta v}{\partial y}\,dxdy+\int\sigma_y m\,\delta v dS$$

and

$$\begin{aligned}&\iint\left(\frac{\partial\tau_{xy}}{\partial x}\,\delta v+\frac{\partial\tau_{xy}}{\partial y}\,\delta u\right)dxdy\\&=-\iint\tau_{xy}\left(\frac{\partial\delta v}{\partial x}+\frac{\partial\delta u}{\partial y}\right)dxdy\\&\quad+\int(\tau_{xy}m\,\delta u+\tau_{xy}l\delta v)\,dS\end{aligned}$$

Thus Eq. (C) is transformed to

$$-\int\int\left[\sigma_x\frac{\partial\delta u}{\partial x}+\sigma_y\frac{\partial\delta v}{\partial y}+\tau_{xy}\left(\frac{\partial\delta v}{\partial x}+\frac{\partial\delta u}{\partial y}\right)\right]dxdy$$
$$+\int\int(X_b\,\delta u+Y_b\,\delta v)\,dxdy+\int(\sigma_x l+\tau_{xy}\,m)\,\delta u \qquad \text{(d)}$$
$$+(\tau_{xy}l+\sigma_y m)\,\delta v]\,dS=0$$

Eq (d) becomes

$$-\delta U+\delta We=0$$

$$\delta We=\delta U$$

For equilibrium to exist the total external virtual work is equal to the total internal virtual work. Thisma the-matical statement is a necessary condition for equilibrium. Again if we apply Green's theorem in the opposite direction

$$\int\int\left\{\left(\frac{\partial\sigma_x}{\partial x}+\frac{\partial\tau_{xy}}{\partial y}+X_b\right)\delta u+\left(\frac{\partial\sigma_y}{\partial y}+\frac{\partial\tau_{xy}}{\partial x}+Y_b\right)\delta v\right\}dxdy$$
$$=\int\{(\sigma_x l+\tau_{xy}\,m-X_s)\,\delta u+(\tau_{xy}l+\sigma_y\,m-Y_s)\,\delta v\}\,dS$$

Thus, the principle of virtual displacement can be stated as that a de-formable system is in equilibrium if the total external virtual work is equal to the total internal virtual work for every virtual displacement satisfying the kinematic boundary conditions. It may be noted here that the principle of virtual displacement is an equilibrium requirement and is independent of the material behaviour.

Shape Function

Based on the shapes elements can be classified as

(i) One dimensional elements

(ii) Two dimensional elements

(iii)Three dimensional elements

One dimensional elements

These elements are suitable for the analysis of one dimensional problem and may be called as line elements also. Figure 1.3 shows different types of one dimensional elements.

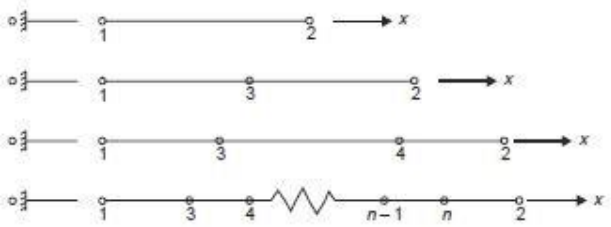


Fig 1.3 one dimensional elements

Two Dimensional elements

We need two dimensional elements to solve two dimensional problems. Common two dimensional problems in stress analysis are plane stress, plane strain and plate problems. Two dimensional elements often used is three noded triangular element shown in Fig. 1.4. It has the distinction of being the first and most used element. These elements are known as Constant Strain Triangles (CST) or Linear Displacement Triangles.

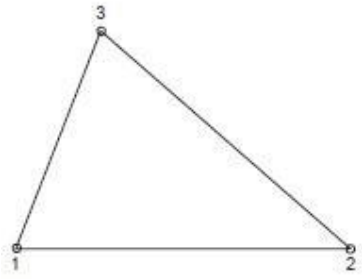


Fig 1.4 Constant Strain triangle

Six noded and ten noded triangular elements (Fig. 1.5) are also used by the analysts. Six noded triangular element is known as Linear Strain Triangle (LST) or as Quadratic Displacement Triangle. Ten noded triangular elements are known as Quadratic Strain Triangles (QST) or Cubic Displacement Triangles. One can think of trying the use of still higher order triangular elements like Cubic Strain Triangles and Quartic Strain Triangles. A simple but less used two dimensional element is the four noded rectangular element whose sides are parallel to the global coordinate systems (Fig. 1.7). This systems is easy to construct automatically but it is not well suited to approximate inclined boundaries.

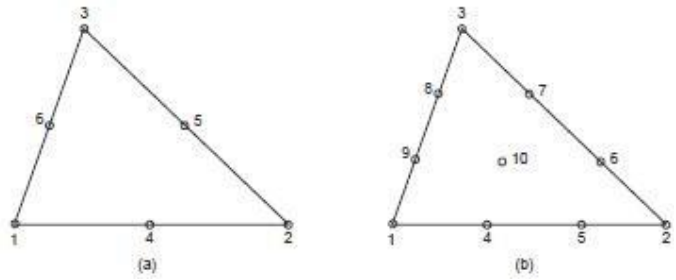


Fig 1.5 (a) Linear strain triangle (b) Quadratic Strain triangle

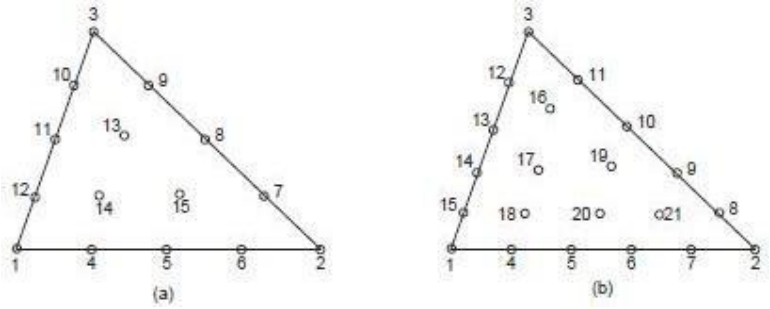


Fig 1.6 (a) Cubic strain triangle (b) Quadratic strain triangle

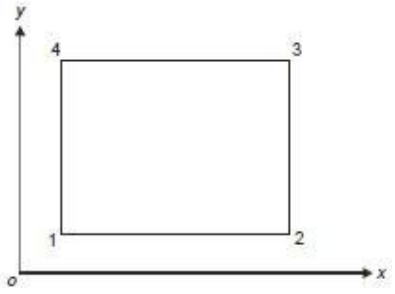


Fig 1.7 4 noded rectangular element

Lagrange, Serendipity and iso parametric Elements

Rectangular elements of higher order also can be used. Figure 1.8 shows a family of Lagrange rectangle in which nodes are in the form of grid points. Figure 1.9 shows the family of Serendipity rectangles which are having nodes only along the external boundaries.

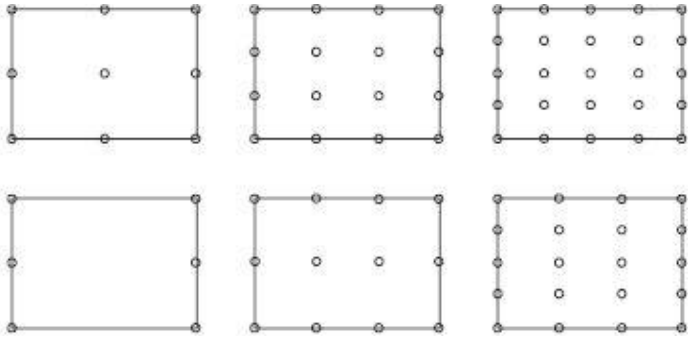


Fig 1.8 Lagrange family rectangular elements

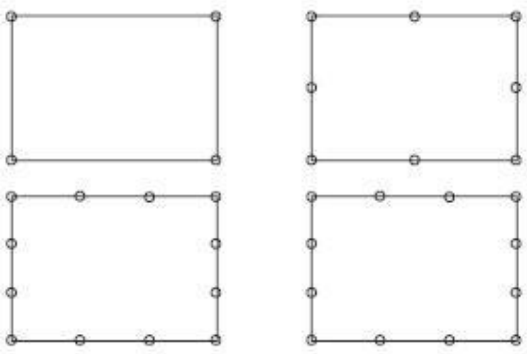


Fig 1.9 Serendipity Family rectangular elememts

Quadrilateral Elements are also used in finite element analysis (Fig. 1.10). Initially quadrilateral elements were developed by combining triangular elements (Fig. 1.11). But it has taken back stage after isoparametric concept was developed. Isoparametric concept is based on using same functions for defining geometries and nodal unknowns. Even higher order triangular elements may be used to generate quadrilateral elements.

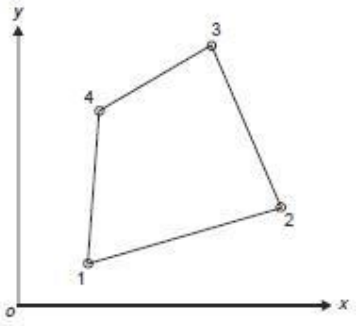


Fig 1.10 Quadrilateral element

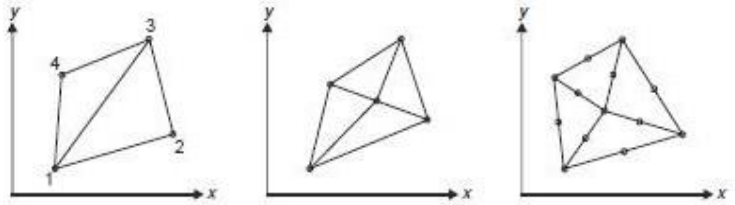


Fig 1.11 Quadrilateral elements generated using triangular elements

Using isoparametric concept even curved elements are developed to take care of boundaries with curved shapes (Fig. 1.12).

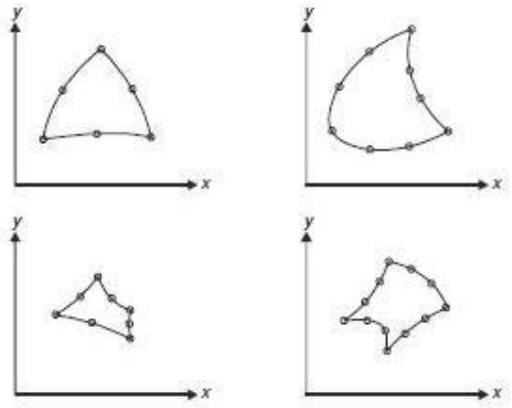


Fig 1.12 iso parametric two dimensional elements

Three Dimensional elements

Similar to the triangle for two dimensional problems tetrahedron is the basic element for three dimensional problems (Fig. 1.13). Tetrahedron is having four nodes, one at each corner. Three dimensional elements with eight nodes are either in the form of a general hexahedron or a rectangular prism, which is a particular case of a hexahedron. The rectangular prism element is many times called as a brick element also. In these elements also one can think of using higher order elements. (Fig. 1.13).

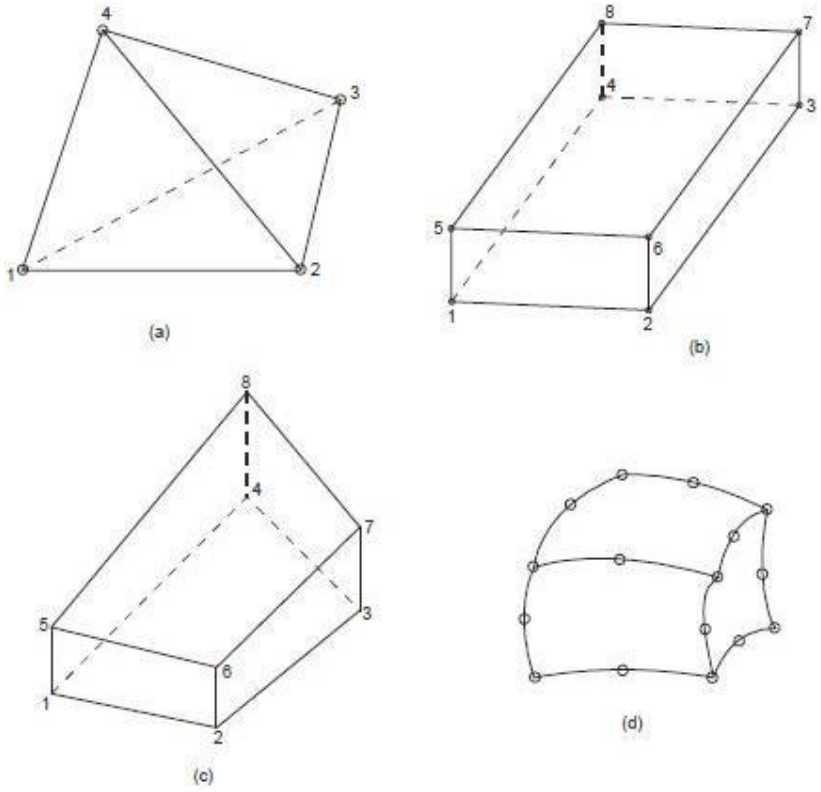


Fig 1.13 (a) tetrahedron element (b) Rectangular prism element
(c) Arbitary hexahedron element (d) Three dimensional quadratic element

Minimum potential energy

The minimum total potential energy principle is a fundamental concept used in physics, chemistry, biology, and engineering. It asserts that a structure or body shall deform or displace to a position that minimizes the total potential energy, with the lost potential energy being dissipated as heat. For example, a marble placed in a bowl will move to the bottom and rest there, and similarly, a tree branch laden with snow will bend to a lower position. The lower position is the position for minimum potential energy: it is the stable configuration for equilibrium. The principle has many applications in structural analysis and solid mechanics.

A binding energy is the energy that must be exported from a system for the system to enter a bound state. If the potential energy is chosen to be zero when the system is unbound, the potential energy of the system is negative after it enters a bound state.[1] A bound system has a lower (*i.e.*, more negative) potential energy than the sum of its parts—this is what keeps the system aggregated in accordance with the minimum total potential energy principle**.**

The total potential energy,π is the sum of the elastic strain energy,U, stored in the deformed body and the potential energy V associated to the applied forces:

$$\pi = U+V$$

This energy is at a stationary position when an infinitesimal variation from such position involves no change in energy:

$$\delta \pi = \delta (U+V) = 0$$

The principle of minimum total potential energy may be derived as a special case of the virtual work principle for elastic systems subject to conservative forces. The equality between external and internal virtual work (due to virtual displacements) is:

$$\int \delta U^{T} T \, ds + \int \delta u^{T} dv = \int \delta \varepsilon^{T} dv$$

Where

U = Vector of displacements

T=Vector of distributed forces on the part St of the surface

F=Vector of body forces

In the special case of elastic bodies, the right-hand-side of can be taken to be the change, δU, of elastic strain energy **U** due to infinitesimal variations of real

displacements. In addition, when the external forces are conservative forces, the left-hand-side of can be seen as the change in the potential energy function **V** of the forces. The function **V** is defined as:

$$V = -\int u^T ds - \int u^T f dv$$

where the minus sign implies a loss of potential energy as the force is displaced in its direction. With these two subsidiary conditions, becomes:

$$-\delta V = \delta U$$

Generation of Stiffness Matrix

The stiffness matrix of a structural system can be derived by various methods like Variational principle, Galerkin method etc. The derivation of an element stiffness matrix has already been discussed in earlier lecture. The stiffness matrix is an inherent property of the structure. Element stiffness is obtained with respect to its axes and then transformed this stiffness to structure axes. The properties of stiffness matrix are as follows:

- Stiffness matrix is symmetric and square.
- In stiffness matrix, all diagonal elements are positive.
- Stiffness matrix is positive definite

For example, if K is a symmetric $n \times n$ real matrix and x is non-zero column vector, then K will be positive definite while $x^T Kx$ is positive.

Global Stiffness Matrix

A structural system is an assemblage of number of elements. These elements are interconnected together to form the whole structure. Therefore, the element stiffness of all the elements are first need to be calculated and then assembled together in systematic manner. It may be noted that the stiffness at a joint is obtained by adding the stiffness of all elements meeting at that joint. To start with, the degrees of freedom of the structure are numbered first. This numbering will start from 1 to n where n is the total degrees of freedom. These numberings are referred to as degrees of freedom corresponding to global degrees of freedom. The element stiffness matrix of each element should be placed in their proper position in the overall stiffness matrix. The following steps may be performed to calculate the global stiffness matrix of the whole structure.

a. Initialize global stiffness matrix [K] as zero. The size of global stiffness matrix will be equal to the total degrees of freedom of the structure.

b. Compute individual element properties and calculate local stiffness matrix [k] of that element.

c. Add local stiffness matrix [k] to global stiffness matrix [k] using proper locations

d. Repeat the Step b. and c. till all local stiffness matrices are placed globally.

The steps to be followed in the computer program are shown in the form of flow chart in Fig. 1.14 for assembling the local stiffness matrix to global stiffness matrix

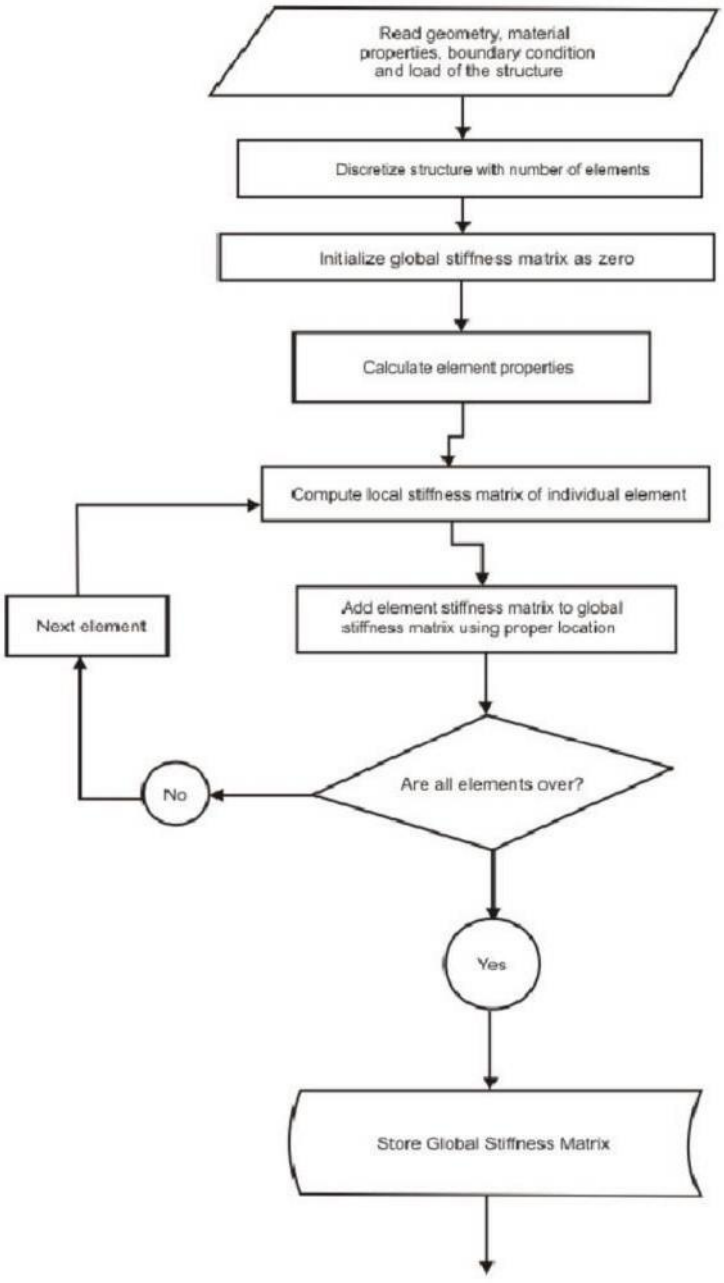


Fig 1.14 Assembly of stiffness matrix from local to global

SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF BUILDING AND ENVIRONMENT

DEPARTMENT OF CIVIL ENGINEERING

UNIT – II - STRESS ANALYSIS – SCIA5201

UNIT-II STRESS ANALYSIS

The problems of solid mechanics may be formulated as three-dimensional problems and finite element technique may be used to solve them. In many practical situations, the geometry and loading will be such that the problems may be formulated to two-dimensional or one-dimensional problems without much loss of accuracy. The relation between strain and displacement for two dimensional problems can be simplified as follows.

$$\varepsilon_x = \frac{\partial u}{\partial x} \quad \text{(a)}$$

$$\varepsilon_y = \frac{\partial u}{\partial x} \quad \text{(b)}$$

$$\gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}$$

The above expression can be written in a combined form: (c)

$$\frac{\partial^2 \varepsilon_x}{\partial y^2} + \frac{\partial^2 \varepsilon_y}{\partial x^2} = \frac{\partial^2 \gamma_{xy}}{\partial x \partial y}$$

Eq. (c) is the compatibility equation since it states the geometric requirements. This condition will ensure adjacent elements to remain free from discontinuities such as gaps and overlaps.

Plane stress problem

The plane stress problem is characterized by very small dimensions in one of the normal directions. Some typical examples are shown in Fig. 1. In these cases, it is assumed that no stress component varies across the thickness and the stress components σ_z, τ_{xz} and τ_{yz} are zero. The state of stress is specified by σ_x, σ_y and τ_{xy} only and is called plane stress.

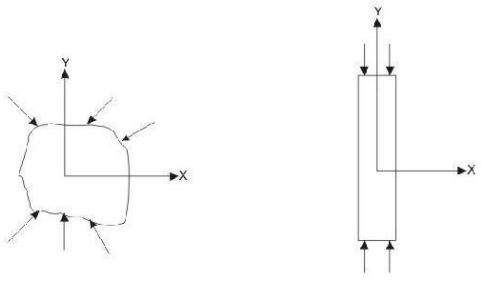


Fig. 1 Plane stress example: Thin plate with in-plane loading

The stress components may be expressed in terms of strain, which is as follows.

$$\begin{Bmatrix}\sigma_x\\ \sigma_y\\ \tau_{xy}\end{Bmatrix}=\frac{E}{1-\mu^2}\begin{bmatrix}1 & \mu & 0\\ \mu & 1 & 0\\ 0 & 0 & \frac{1-\mu}{2}\end{bmatrix}\begin{Bmatrix}\varepsilon_x\\ \varepsilon_y\\ \gamma_{xy}\end{Bmatrix}-\frac{E\alpha T}{1-\mu}\begin{Bmatrix}1\\ 1\\ 0\end{Bmatrix}$$

The strain components can also be expressed in terms of the stress, which is given below

$$\begin{Bmatrix}\varepsilon_x\\ \varepsilon_y\\ \gamma_{xy}\end{Bmatrix}=\frac{1}{E}\begin{bmatrix}1 & -\mu & 0\\ -\mu & 1 & 0\\ 0 & 0 & 2(1+\mu)\end{bmatrix}\begin{Bmatrix}\sigma_x\\ \sigma_y\\ \tau_{xy}\end{Bmatrix}+\alpha T\begin{Bmatrix}1\\ 1\\ 0\end{Bmatrix}$$

It can also be shown that

$$\varepsilon_z=\frac{-\mu}{1-\mu}(\varepsilon_x+\varepsilon_y)+\frac{1+\mu}{1-\mu}\alpha T \text{ and } \quad \gamma_{yz}=\gamma_{zx}=0$$

Plane strain problem

Problems involving long bodies whose geometry and loading do not vary significantly in the longitudinal direction are referred to as plane strain problems. Some typical examples are given in Fig. 2. In these cases, a constant longitudinal displacement corresponding to a rigid body translation and displacements linear in z corresponding to rigid body rotation do not result in strain. As a result, the following relations arise.

$$\varepsilon_z=\gamma_{yz}=\gamma_{zx}=0$$

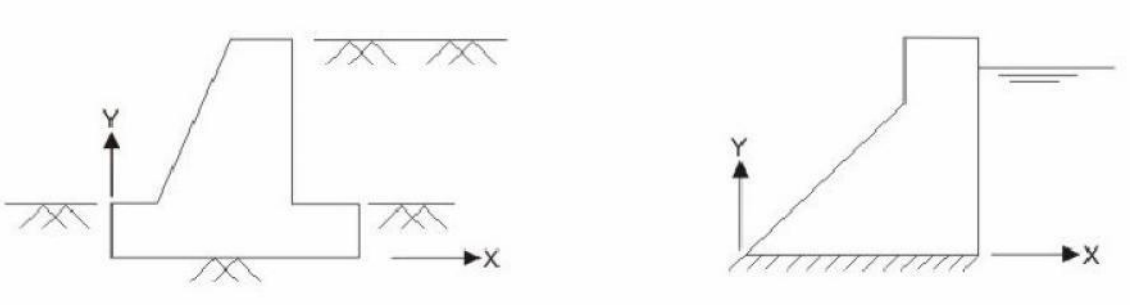
The constitutive relation for elastic isotropic material for this case may be given by

$$\begin{Bmatrix}\sigma_x\\ \sigma_y\\ \tau_{xy}\end{Bmatrix}=\frac{E}{(1+\mu)(1-2\mu)}\begin{bmatrix}1-\mu & \mu & 0\\ \mu & 1-\mu & 0\\ 0 & 0 & \frac{1-2\mu}{2}\end{bmatrix}\begin{Bmatrix}\varepsilon_x\\ \varepsilon_y\\ \gamma_{xy}\end{Bmatrix}-\frac{E\alpha T}{1-2\mu}\begin{Bmatrix}1\\ 1\\ 0\end{Bmatrix}$$

$$\text{Also} \quad \sigma_z=\mu(\sigma_x+\sigma_y)-E\alpha T \text{ and } \quad \tau_{yz}=\tau_{zx}=0$$

The strain components can be expressed in terms of the stress as follows.

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \frac{(1+\mu)}{E} \begin{bmatrix} (1-\mu) & -\mu & 0 \\ -\mu & (1-\mu) & 0 \\ 0 & 0 & 2 \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} + (1+\mu)\alpha T \begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix}$$



(a) Retaining wall **(b) Dam**

Fig. 2 Plane strain examples

Axisymmetric Problem

Many problems in stress analysis which are of practical interest involve solids of revolution subject to axially symmetric loading. A circular cylinder loaded by a uniform internal or external pressure, circular footing resting on soil mass, pressure vessels, rotating wheels, flywheels etc. The strain displacement relations in these type of problems are given by

$$\varepsilon_x = \frac{\partial u}{\partial x}$$

$$\varepsilon_\theta = \frac{u}{x}$$

$$\varepsilon_y = \frac{\partial v}{\partial y}$$

$$\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}$$

The two components of displacements in any plane section of the body along its axis of symmetry define completely the state of strain and therefore the state of stress. The constitutive relations are given below for such types of problems.

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_\theta \\ \tau_{xy} \end{Bmatrix} = \frac{E}{(1+\mu)(1-2\mu)} \begin{bmatrix} (1-\mu) & \mu & \mu & 0 \\ \mu & 1-\mu & \mu & 0 \\ \mu & \mu & 1-\mu & 0 \\ 0 & 0 & 0 & \frac{1-2\mu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_\theta \\ \gamma_{xy} \end{Bmatrix}$$

Triangular element

The triangular element can be used to represent the arbitrary geometry much easily. On the other hand, rectangular elements, in general, are of limited use as they are not well suited for representing curved boundaries. However, an assemblage of rectangular and triangular element with triangular elements near the boundary can be very effective (Fig. 3).

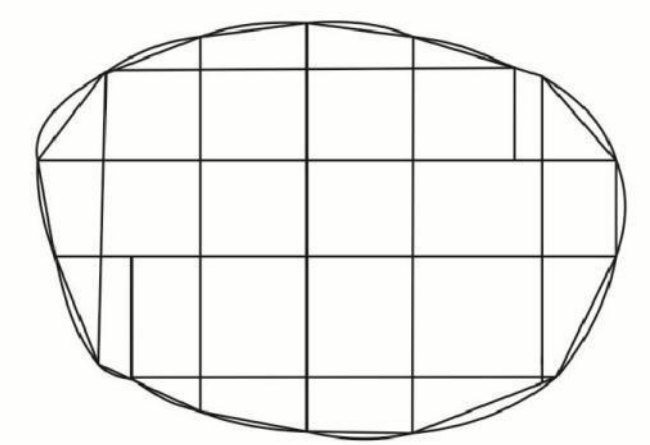


Fig. 3 Finite element mesh consisting of triangular and rectangular element

The shape function for triangular elements (linear, quadratic and cubic) with various nodes (Fig. 4) can be formulated. An internal node will exist for cubic element as seen in Fig. 4(c).

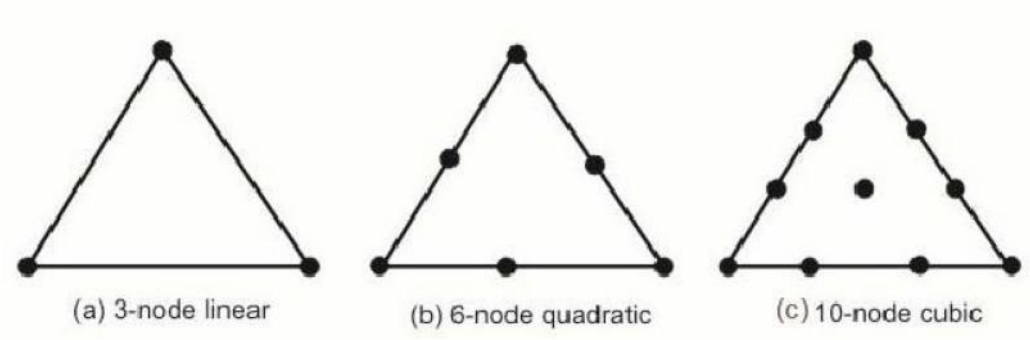


Fig. 4 Triangular elements

In displacement formulation, it is very important to approximate the variation of displacement in the element by suitable function. The interpolation function can be derived either using the Cartesian coordinate system or by the area coordinates.

Shape function using Cartesian coordinates

Polynomials are easiest way of mathematical operation for expressing variation of displacement. For example, the displacement variation within the element can be represented by the following function in case of two dimensional plane stress/strain problems.

$$u = \alpha_0 + \alpha_1 x + \alpha_2 y$$

$$v = \alpha_3 + \alpha_4 x + \alpha_5 y$$

where α_0, α_1, α_2 ….. are unknown coefficients. Thus the displacement vectors at any point *P*,in the element (Fig.4) can be expressed with the following relation.

$$\{d\} = \begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y \end{bmatrix} \begin{Bmatrix} \alpha_0 \\ \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \\ \alpha_5 \end{Bmatrix}$$

Or, $\{d\}=[\phi]\{\alpha\}$

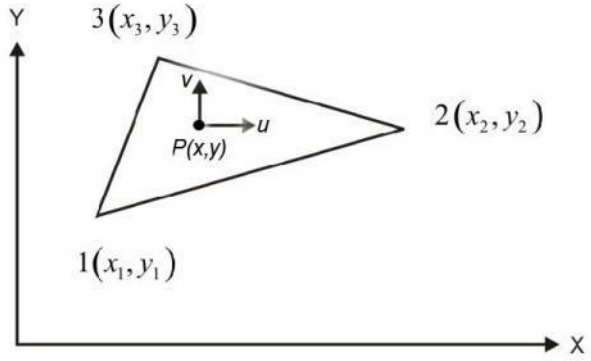


Fig. 4 Triangular element in Cartesian Coordinates

Similarly, for "*m*" node element having three degrees of freedom at each node, the displacement function can be expressed as

$$u = \alpha_0 + \alpha_1 x + \alpha_2 y + \alpha_3 x^2 + \alpha_4 xy + \alpha_5 y^2 + \ldots\ldots + \alpha_{m-1} y^n$$
$$v = \alpha_m + \alpha_{m+1} x + \alpha_{m+2} y + \alpha_{m+3} x^2 + \alpha_{m+4} xy + \ldots\ldots + \alpha_{2m-1} y^n$$
$$w = \alpha_{2m} + \alpha_{2m+1} x + \alpha_{2m+2} y + \alpha_{2m+3} x^2 + \alpha_{2m+4} xy + \ldots\ldots + \alpha_{3m-1} y^n$$

Hence, in such case,

$$\{d\} = \begin{Bmatrix} u \\ v \\ w \end{Bmatrix} = \begin{bmatrix} \{\phi\}^T & 0 & 0 \\ 0 & \{\phi\}^T & 0 \\ 0 & 0 & \{\phi\}^T \end{bmatrix} \{\alpha\}$$

Where, $\{\alpha\}^T = [\alpha_0 \alpha_1 \ldots \alpha_{3m-1}]$ and, $[\phi]^T = [1 \quad x \quad y \quad x^2 \quad xy \ldots\ldots \quad y^n]$

Now, for a linear triangular element with 2 degrees of freedom, eq. (3.2.3) can be written in terms of the nodal displacements as follows.

$$\{d\} = \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ v_1 \\ v_2 \\ v_3 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 & 0 & 0 & 0 \\ 1 & x_2 & y_2 & 0 & 0 & 0 \\ 1 & x_3 & y_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x_1 & y_1 \\ 0 & 0 & 0 & 1 & x_2 & y_2 \\ 0 & 0 & 0 & 1 & x_3 & y_3 \end{bmatrix} \begin{Bmatrix} \alpha_0 \\ \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \\ \alpha_5 \end{Bmatrix}$$

Where, $\{d\}$ is the nodal displacements. To simplify the above expression for finding out the shape function, the displacements in X direction can be separated out which will be as follows:

$$\{u_i\}=\begin{Bmatrix}u_1\\u_2\\u_3\end{Bmatrix}=\begin{bmatrix}1 & x_1 & y_1\\1 & x_2 & y_2\\1 & x_3 & y_3\end{bmatrix}\begin{Bmatrix}\alpha_0\\\alpha_1\\\alpha_2\end{Bmatrix}$$

To obtain the polynomial coefficients, $\{\alpha\}$ the matrix of the above equation are to be inverted. Thus,

$$\begin{Bmatrix}\alpha_0\\\alpha_1\\\alpha_2\end{Bmatrix}=\begin{bmatrix}1 & x_1 & y_1\\1 & x_2 & y_2\\1 & x_3 & y_3\end{bmatrix}^{-1}\begin{Bmatrix}u_1\\u_2\\u_3\end{Bmatrix}=\frac{1}{2A}\begin{bmatrix}x_2y_3-x_3y_2 & x_3y_1-x_1y_3 & x_1y_2-x_2y_1\\y_2-y_3 & y_3-y_1 & y_1-y_2\\x_3-x_2 & x_1-x_3 & x_2-x_1\end{bmatrix}\begin{Bmatrix}u_1\\u_2\\u_3\end{Bmatrix}$$

$$=\frac{1}{2A}\begin{bmatrix}a_1 & a_2 & a_3\\b_1 & b_2 & b_3\\c_1 & c_2 & c_3\end{bmatrix}\begin{Bmatrix}u_1\\u_2\\u_3\end{Bmatrix}$$

Where, A is the area of the triangle and can be obtained as follows.

$$A=\frac{1}{2}\begin{vmatrix}1 & x_1 & y_1\\1 & x_2 & y_2\\1 & x_3 & y_3\end{vmatrix}$$

$$u=\frac{1}{2A}\left[(x_2y_3-x_3y_2)+(y_2-y_3)x+(x_3-x_2)y\right]u_1$$
$$+\frac{1}{2A}\left[(x_3y_1-x_1y_3)+(y_3-y_1)x+(x_1-x_3)y\right]u_2$$
$$+\frac{1}{2A}\left[(x_1y_2-x_2y_1)+(y_1-y_2)x+(x_2-x_1)y\right]u_3$$

Thus, the interpolation function can be obtained from the above as:

$$\{N\}^T = \begin{Bmatrix} N_1 \\ N_2 \\ N_3 \end{Bmatrix} = \begin{Bmatrix} \frac{1}{2A}\left[(x_2y_3 - x_3y_2) + (y_2 - y_3)x + (x_3 - x_2)y\right] \\ \frac{1}{2A}\left[(x_3y_1 - x_1y_3) + (y_3 - y_1)x + (x_1 - x_3)y\right] \\ \frac{1}{2A}\left[(x_1y_2 - x_2y_1) + (y_1 - y_2)x + (x_2 - x_1)y\right] \end{Bmatrix}$$

Such three node triangular element is commonly known as constant strain triangle (CST) as its strain is assumed to be constant inside the element.

CST is the simplest element to develop mathematically. As there is no variation of strain inside the element, the mesh size of the triangular element should be small enough to get correct results. This element produces constant temperature gradients ensuring constant heat flow within the element for heat transfer problems.

Shape function for six node element

Fig. 5 shows a triangular element with six nodes. The additional three nodes (4, 5, and 6) are situated at the midpoints of the sides of the element. A complete polynomial representation of the field variable can be expressed with the help of Pascal triangle:

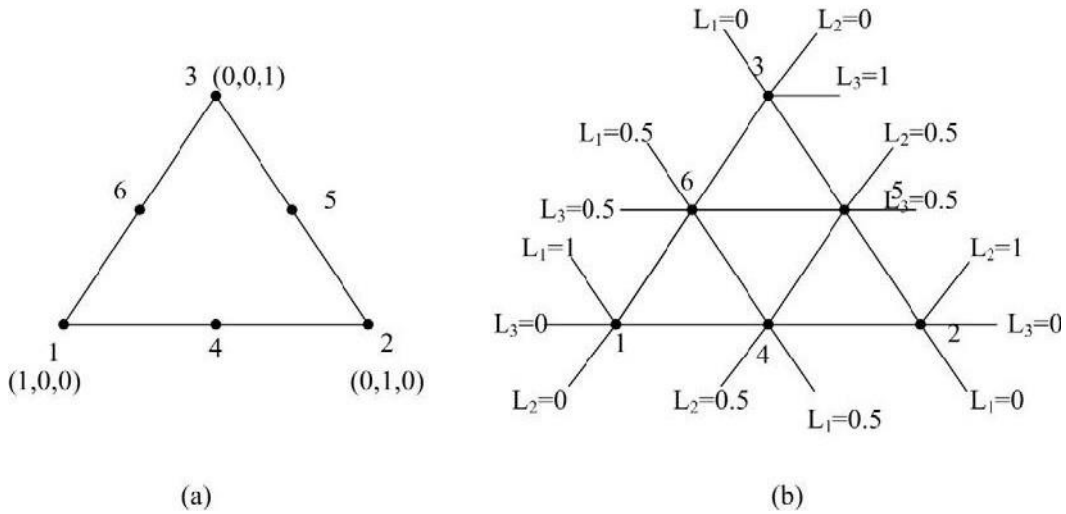


Fig. 5 (a) Six node triangular element (b) Lines of constant values of the area coordinates

Using the above field variable function, one can reach the following expression using interpolation function and the nodal values.

$$\emptyset(x,y) = \alpha_0 + \alpha_1 x + \alpha_2 y + \alpha_3 x^2 + \alpha_4 xy + \alpha_5 y^2$$

$$\phi(x,y) = \sum_{i=1}^{6} N_i(x,y)\phi_i$$

Here, the every shape function must be such that its value will be unity if evaluated at its related node and zero if evaluated at any of the other five nodes. Moreover, as the field variable representation is quadratic, each interpolation function will also become quadratic. Fig. 5 (a) shows the six node element with node numbering convention along with the area coordinates at three corners. The six node element with lines of constant values of the area coordinates passing through the nodes is shown in Fig. 5 (b). Now the interpolation functions can be constructed with the help of area coordinates from the above diagram. For example, the interpolation function *N1* should be unity at node 1 and zero at all other five nodes. According to the above diagram, the value of *L1* is 1 at node 1 and ½ at node 4 and 6. Again, *L1* will be 0 at nodes 2, 3 and 5. To satisfy all these conditions, one can propose following expression:

$$N_1(X,Y) = N_1(L_1, L_2, L_3) = L1\left[L1 - \frac{1}{2}\right]$$

Evaluating the above expression, the value of *N1* is becoming ½ at node 1 though it must become unity. Therefore, the above expression is slightly modified satisfying all the conditions and will be as follows:

$$N_1 = 2L_1\left[L_1 - \frac{1}{2}\right] = L_1(2L_1 - 1)$$

This assures the required conditions at all the six nodes and is a quadratic function, as*L*1 is a linear function of *x* and *y*. The remaining five interpolation functions can also be obtained in similar fashion applying the required nodal conditions. Thus, the shape function for the six node triangle element can be written as given below.

$$N_1 = L_1(2L_1 - 1)$$
$$N_2 = L_2(2L_2 - 1)$$
$$N_3 = L_3(2L_3 - 1)$$
$$N_4 = 4L_1L_2$$
$$N_5 = 4L_2L_3$$
$$N_6 = 4L_3L_1$$

Such six node triangular element is commonly known as linear strain triangle (LST) as its strain is assumed to vary linearly inside the element. In case of 2-D plane stress/strain problem, the element displacement field for such quadratic triangle may be expressed as so the element strain can be derived from the above displacement field as follows.

$$u(x,y) = \alpha_0 + \alpha_1 x + \alpha_2 y + \alpha_3 x^2 + \alpha_4 xy + \alpha_5 y^2$$
$$v(x,y) = \alpha_6 + \alpha_7 x + \alpha_8 y + \alpha_9 x^2 + \alpha_{10} xy + \alpha_{11} y^2$$

So the element strain can be derived from the above displacement field as follows.

$$\varepsilon_x = \frac{\partial u}{\partial x} = \alpha_1 + 2\alpha_3 x + \alpha_4 y$$
$$\varepsilon_y = \frac{\partial v}{\partial y} = \alpha_8 + \alpha_{10} x + 2\alpha_{11} y$$
$$\gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} = \alpha_2 + \alpha_4 x + 2\alpha_5 y + \alpha_7 + 2\alpha_9 x + \alpha_{10} y$$

The above expression shows that the strain components are linearly varying inside the element. Therefore, this six node element is called linear strain triangle. The main advantage of this element is that it can capture the variation of strains and therefore stresses of the element.

Construction of Shape Function by Degrading Technique

Sometimes, the geometry of the structure or its loading and boundary conditions are such that the stresses developed in few locations are quite high. On the other hand, variations of stresses are less in some areas and as a result, refinement of finite element mesh is not necessary. It would be economical in terms of computation if higher order elements are chosen where stress concentration is high and lower order elements at area away from the critical area. Fig.6 shows graphical representations where various order of triangular elements are used for generating a finite element mesh.

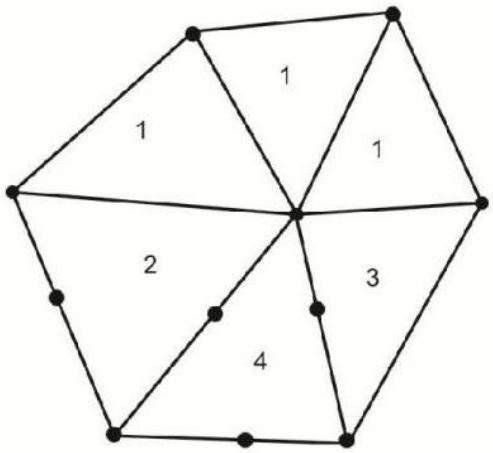


Fig. 6 Triangular elements with different number of nodes

Fig. 6 contains four types of element. Type 1 has only three nodes, type 2 element has five nodes, type 3 has four nodes and type 4 has six nodes. The shape function for 3-node and 6-node triangular elements has already been derived. The shape functions of 6-node element can suitably be degraded to derive shape functions of other two types of triangular elements.

Quadrilateral element

Shape Function

The shape function of eight node rectangular element can be derived in similar fashion as done in case of four node element. The only difference will be on choosing of polynomial as this element is of quadratic in nature. The derivation will be algebraically complex in case of using Cartesian coordinate system. However, use of the natural coordinate system will make the process simpler as

the natural coordinates vary from -1 to +1 in the element. The variation of filed variable ϕ can be expressed in natural coordinate system by the following polynomial.

$$\phi(\xi,\eta)=\alpha_0+\alpha_1\xi+\alpha_2\eta+\alpha_3\xi^2+\alpha_4\xi\eta+\alpha_5\eta^2+\alpha_6\xi^2\eta+\alpha_7\xi\eta^2$$

It may be noted that the cubic terms ξ3 and η3are omitted and geometric invariance is ensured by choosing the above expression. Fig. 8 shows the natural nodal coordinates of the eight node rectangle element in natural coordinate system.

The nodal field variables can be obtained from the above expression after putting the coordinates at nodes.

$$\{\phi_i\}=\begin{Bmatrix}\phi_1\\ \phi_2\\ \phi_3\\ \phi_4\\ \phi_5\\ \phi_6\\ \phi_7\\ \phi_8\end{Bmatrix}=\begin{bmatrix}1&-1&-1&1&1&1&-1&-1\\ 1&1&-1&1&-1&1&-1&1\\ 1&1&1&1&1&1&1&1\\ 1&-1&1&1&-1&1&1&-1\\ 1&0&-1&0&0&1&0&0\\ 1&1&0&1&0&0&0&0\\ 1&0&1&0&0&1&0&0\\ 1&-1&0&1&0&0&0&0\end{bmatrix}\begin{Bmatrix}\alpha_0\\ \alpha_1\\ \alpha_2\\ \alpha_3\\ \alpha_4\\ \alpha_5\\ \alpha_6\\ \alpha_7\end{Bmatrix}=[A]\{\alpha_i\}$$

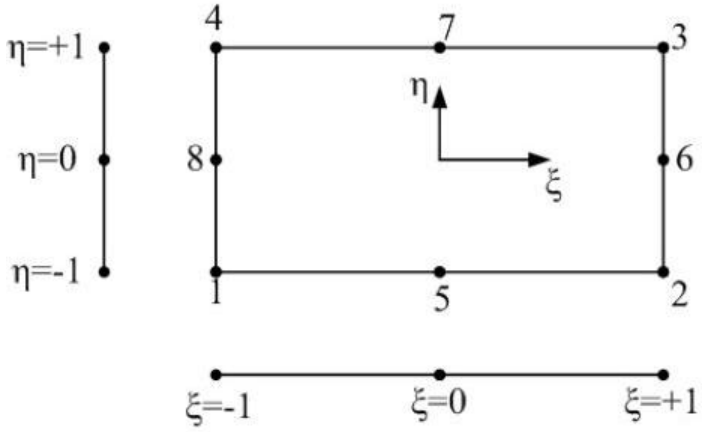


Fig. 8 Natural coordinates of eight node rectangular element

Replacing the unknown coefficient α_i

$$\phi(\xi,\eta)=\begin{bmatrix}1 & \xi & \eta & \xi^2 & \xi\eta & \eta^2 & \xi^2\eta & \xi\eta^2\end{bmatrix}[A]^{-1}\{\phi_i\}$$

$$=\begin{bmatrix}1 & \xi & \eta & \xi^2 & \xi\eta & \eta^2 & \xi^2\eta & \xi\eta^2\end{bmatrix}\frac{1}{4}\begin{bmatrix}-1 & -1 & -1 & -1 & 2 & 2 & 2 & 2\\ 0 & 0 & 0 & 0 & 0 & 2 & 0 & -2\\ 0 & 0 & 0 & 0 & -2 & 0 & 2 & 0\\ 1 & 1 & 1 & 1 & -2 & 0 & -2 & 0\\ 1 & -1 & 1 & -1 & 0 & 0 & 0 & 0\\ 1 & 1 & 1 & 1 & 0 & -2 & 0 & -2\\ -1 & -1 & 1 & 1 & 2 & 0 & -2 & 0\\ -1 & 1 & 1 & -1 & 0 & -2 & 0 & 2\end{bmatrix}\begin{Bmatrix}\phi_1\\ \phi_2\\ \phi_3\\ \phi_4\\ \phi_5\\ \phi_6\\ \phi_7\\ \phi_8\end{Bmatrix}$$

$$=\begin{bmatrix}N_1 & N_2 & N_3 & N_4 & N_5 & N_6 & N_7 & N_8\end{bmatrix}\begin{Bmatrix}\phi_1\\ \phi_2\\ \phi_3\\ \phi_4\\ \phi_5\\ \phi_6\\ \phi_7\\ \phi_8\end{Bmatrix}$$

Thus, the interpolation function will become

$$N_1=\frac{(1-\xi)(1-\eta)(-\xi-\eta-1)}{4};\quad N_2\frac{(1+\xi)(1-\eta)(\xi-\eta-1)}{4};$$

$$N_3=\frac{(1+\xi)(1+\eta)(\xi+\eta-1)}{4};N_4=\frac{(1-\xi)(1+\eta)(-\xi+\eta-1)}{4};$$

$$N_5=\frac{(1+\xi)(1-\xi)(1-\eta)}{2};\ N_6=\frac{(1+\xi)(1+\eta)(1-\eta)}{2};$$

$$N_7=\frac{(1+\xi)(1-\xi)(1+\eta)}{2};\ N_8=\frac{(1-\xi)(1+\eta)(1-\eta)}{2}$$

The shape functions of rectangular elements with higher nodes can be derived in similar manner using appropriate polynomial satisfying all necessary criteria. However, difficulty arises due to the inversion of large size of the matrix because of higher degree of polynomial chosen. In next lecture, the shape functions of rectangular element with higher nodes will be derived in a much simpler way.

Natural coordinates

Natural coordinate system is basically a local coordinate system which allows the specification of a point within the element by a set of dimensionless numbers whose magnitude never exceeds unity. This coordinate system is found to be very effective in formulating the element properties in finite element formulation. This system is defined in such that the magnitude at nodal points will have unity or zero or a convenient set of fractions. It also facilitates the integration to calculate element stiffness.

The natural coordinate system for a triangular element is generally called as triangular coordinate system. The coordinate of any point *P*inside the triangle is *x,y* in Cartesian coordinate system. Here, three coordinates, *L1*, *L2* and *L3* can be used to define the location of the point in terms of natural coordinate system. The point *P* can be defined by the following set of area coordinates:

$$L_1 = \frac{A_1}{A} \; ; L_2 = \frac{A_2}{A} \; ; L_3 = \frac{A_3}{A}$$

Where,

A_1= Area of the triangle P23
A_2= Area of the triangle P13
A_3= Area of the triangle P12
A=Area of the triangle 123

Thus,

$$A = A_1+A_2+A_3$$

And

$$L_1+L_2+L_3 = 1$$

Therefore, the natural coordinate of three nodes will be: node 1 (1,0,0); node 2 (0,1,0); and node 3 (0,0,1).

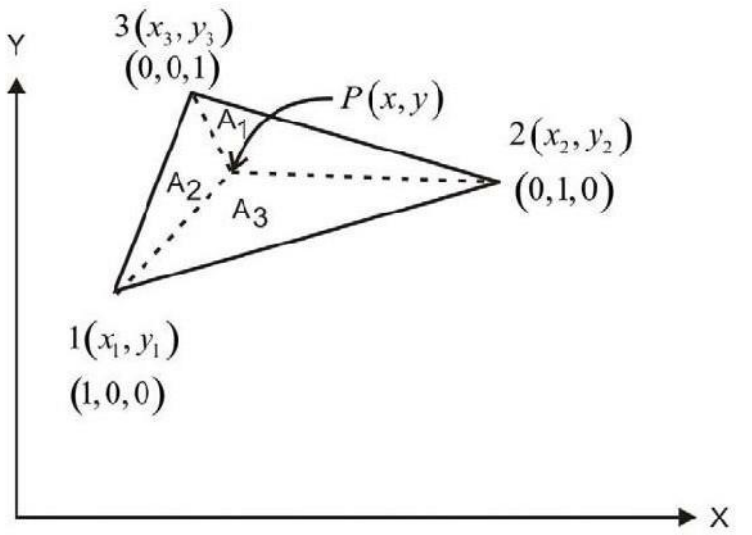


Fig. 9Triangular coordinate system

The area of the triangles can be written using Cartesian coordinates considering *x*, *y* as coordinates of an arbitrary point *P* inside or on the boundaries of the element:

$$A = \frac{1}{2}\begin{vmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{vmatrix}$$

$$A_1 = \frac{1}{2}\begin{vmatrix} 1 & x & y \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{vmatrix}$$

$$A_2 = \frac{1}{2}\begin{vmatrix} 1 & x & y \\ 1 & x_3 & y_3 \\ 1 & x_1 & y_1 \end{vmatrix}$$

$$A_3 = \frac{1}{2}\begin{vmatrix} 1 & x & y \\ 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \end{vmatrix}$$

The relation between two coordinate systems to define point P can be established by their nodal coordinates as

$$\begin{bmatrix}1\\x\\y\end{bmatrix}=\begin{bmatrix}1 & 1 & 1\\x_1 & x_2 & x_3\\y_1 & y_2 & y_3\end{bmatrix}\begin{bmatrix}L_1\\L_2\\L_3\end{bmatrix}$$

Where,

$$x = L_1x_1 + L_2x_2 + L_3x_3$$

$$y = L_1y_1 + L_2y_2 + L_3y_3$$

The inverse between natural and Cartesian coordinates may be expressed as:

$$\begin{bmatrix}L_1\\L_2\\L_3\end{bmatrix}=\frac{1}{2A}\begin{bmatrix}x_2y_3-x_3y_2 & y_2-y_3 & x_3-x_2\\x_3y_1-x_1y_3 & y_3-y_1 & x_1-x_3\\x_1y_2-x_2y_1 & y_1-y_2 & x_2-x_1\end{bmatrix}\begin{bmatrix}1\\x\\y\end{bmatrix}$$

The derivatives with respect to global coordinates are necessary to determine the properties of an element. The relationship between two coordinate systems may be computed by using the chain rule of partial differentiation as

$$\begin{aligned}\frac{\partial}{\partial x}&=\frac{\partial}{\partial L_1}.\frac{\partial L_1}{\partial x}+\frac{\partial}{\partial L_2}.\frac{\partial L_2}{\partial x}+\frac{\partial}{\partial L_3}.\frac{\partial L_3}{\partial x}\\&=\frac{b_1}{2A}.\frac{\partial}{\partial L_1}+\frac{b_2}{2A}.\frac{\partial}{\partial L_2}+\frac{b_3}{2A}.\frac{\partial}{\partial L_3}\\&=\sum_{i=1}^{3}\frac{b_i}{2A}.\frac{\partial}{\partial L_i}\end{aligned}$$

Where, $b1 = y2 - y3$; $b2 = y3 - y1$ and $b3 = y1 - y2$. Similarly, following relation can be obtained.

$$\frac{\partial}{\partial y}=\sum_{i=1}^{3}\frac{c_i}{2A}.\frac{\partial}{\partial L_i}$$

Where, $c1 = x3 - x2$; $c2 = x1 - x3$ and $c3 = x2 - x1$. The above expressions are looked cumbersome. However, the main advantage is the ease with which polynomial terms can be integrated using following area integral expression.

$$\int_A L_1^p L_2^q L_3^r dA = \frac{p!q!r!}{(p+q+r+2)!} 2A$$

Where 0! is defined as unity.

Shape Function using Area Coordinates

The interpolation functions for the triangular element are algebraically complex if expressed in Cartesian coordinates. Moreover, the integration required to obtain the element stiffness matrix is quite cumbersome. This will be discussed in details in next lecture. The interpolation function and subsequently the required integration can be obtained in a simplified manner by the concept of area coordinates. Considering a linear displacement variation of a triangular element as shown in Fig. 10, the displacement at any point can be written in terms of its area coordinates.

$$u = \alpha_1 L_1 + \alpha_2 L_2 + \alpha_3 L_3$$

Or, $u = \{\phi\}^T \{\alpha\}$

where, $\{\phi\}^T = [L_1 \quad L_2 \quad L_3]$ and $\{\alpha\}^T = \{\alpha_1 \quad \alpha_2 \quad \alpha_3\}$

And $L_1 = \frac{A_1}{A}$; $L_2 = \frac{A_2}{A}$; $L_3 = \frac{A_3}{A}$

Here, A is the total area of the triangle. It is important to note that the area coordinates are dependent as $L_1+L_2+L_3 = 1$. It may be seen from figure that at node 1, $L1 = 1$ while $L2 = L3 = 0$. Similarly for other two nodes: at node 2, $L2 = 1$ while $L1 = L3 = 0$, and $L3 = 1$ while $L2 = L1 = 0$. Now, substituting the area coordinates for node 1, 2 and 3, the displacement components at nodes can be written as

$$\{u_i\} = \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \{\alpha\}$$

Thus, from the above expression, one can obtain the unknown coefficient:

$$\{\alpha\} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}$$

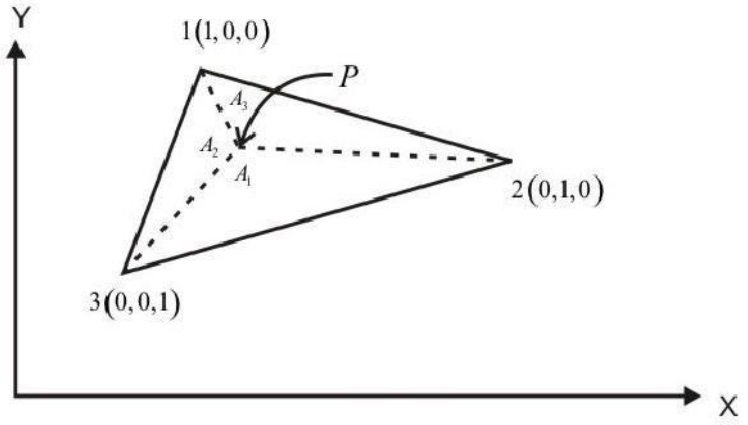


Fig .10 Area coordinates for triangular element

$$\{u\} = \{\phi\}^T \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \{\phi\}^T \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \{u_i\}$$

The above expression can be written in terms of interpolation function as $\{N\}^T \{u\}_i$ Where,

$$\{N\}^T = [L_1 \quad L_2 \quad L_3] \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} = [L_1 \quad L_2 \quad L_3]$$

Similarly, the displacement variation *v* in Y direction can be expressed as follows.

$$v = \{N\}^T \{v_i\}$$

Thus, for two displacement components *u* and *v* of any point inside the element can be written as:

$$\{d\} = \begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} \{N\}^T & \{0\}^T \\ \{0\}^T & \{N\}^T \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \end{Bmatrix}$$

Thus, the shape function of the element will become

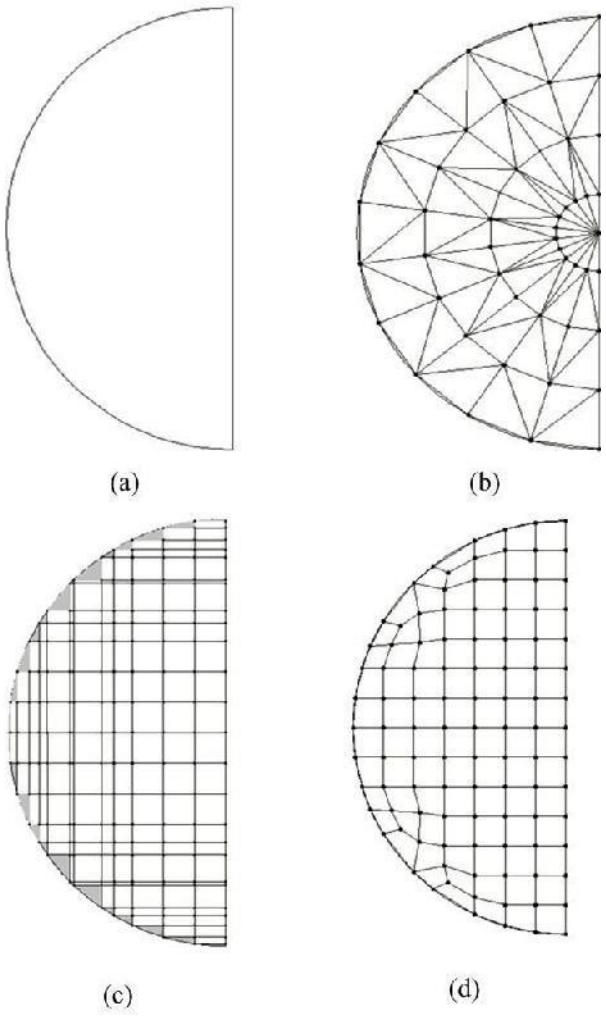
$$[\mathrm{N}] = \begin{bmatrix} \mathrm{L}_1 & \mathrm{L}_2 & \mathrm{L}_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & \mathrm{L}_1 & \mathrm{L}_2 & \mathrm{L}_3 \end{bmatrix}$$

It is important to note that the shape function *Ni* become unity at node *i* and zero at other nodes of the element. The displacement at any point of the element can be expressed in terms of its nodal displacement and the interpolation function as given below.

$$u = N_1 u_1 + N_2 u_2 + N_3 u_3$$
$$v = N_1 v_1 + N_2 v_2 + N_3 v_3$$

Isoparametric Formulation

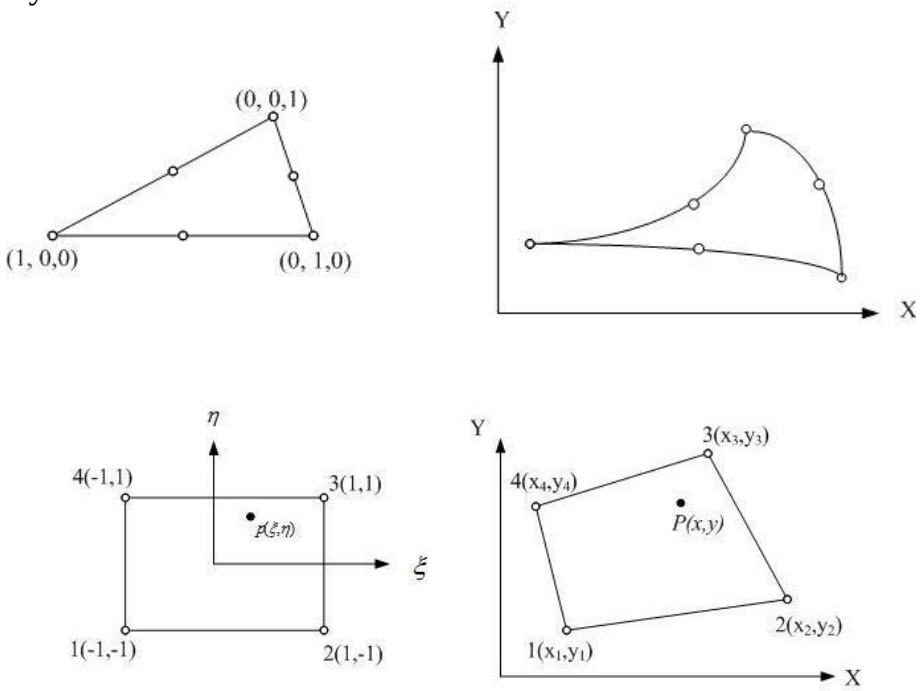
The two or three dimensional elements discussed till now are of regular geometry (e.g. triangular and rectangular element) having straight edge. Hence, for the analysis of any irregular geometry, it is difficult to use such elements directly. For example, the continuum having curve boundary as shown in the Fig. 11 (a) has been discretized into a mesh of finite elements in three ways as shown.

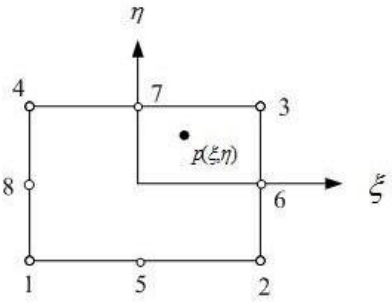


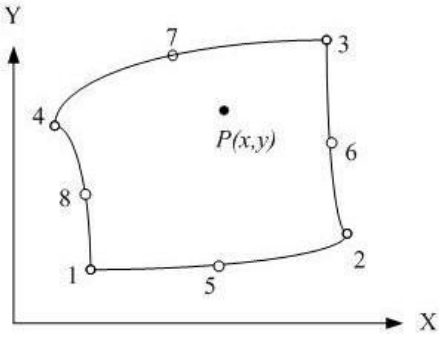
(a) The Continuum to be discritized (b) Discritization using Triangular Elements (c) Discritization using rectangular elements (d) Discritization using a combination of rectangular and quadrilateral elements

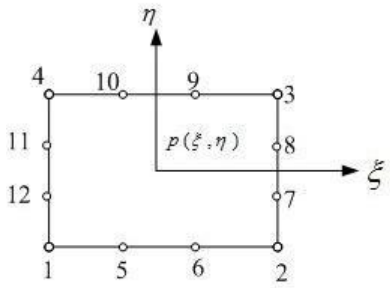
Fig. 11 Discretization of a continuum using different elements

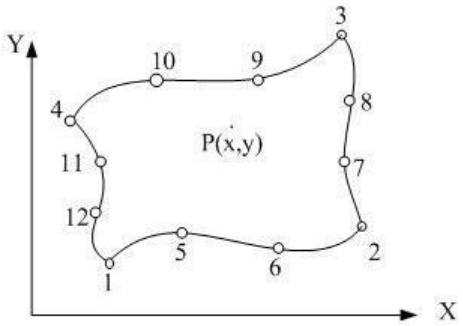
Figure 11 (b) presents a possible mesh using triangular elements. Though, triangular elements can suitable approximate the circular boundary of the continuum, but the elements close to the center becomes slender and hence affect the accuracy of finite element solutions. One possible solution to the problem is to reduce the height of each row of elements as we approach to the center. But, unnecessary refining of the continuum generates relatively large number of elements and thus increases computation time. Alternatively, when meshing is done using rectangular elements as shown in Fig 11 (c), the area of continuum excluded from the finite element model is significantly adequate to provide incorrect results. In order to improve the accuracy of the result one can generate mesh using very small elements. But, this will significantly increase the computation time. Another possible way is to use a combination of both rectangular and triangular elements as discussed below. But such types of combination may not provide the best solution in terms of accuracy, since different order polynomials are used to represent the field variables for different types of elements. Also the triangular elements may be slender and thus can affect the accuracy. In Fig. 11 (d), the same continuum is discritized with rectangular elements near center and with four node quadrilateral elements near boundary. This four-node quadrilateral element can be derived from rectangular elements using the concept of mapping. Using the concept of mapping regular triangular, rectangular or solid elements in natural coordinate system (known as parent element) can be transformed into global Cartesian coordinate system having arbitrary shapes (with curved edge or surfaces). Fig. 12 shows the parent elements in natural coordinate system and the mapped elements in global Cartesian system.

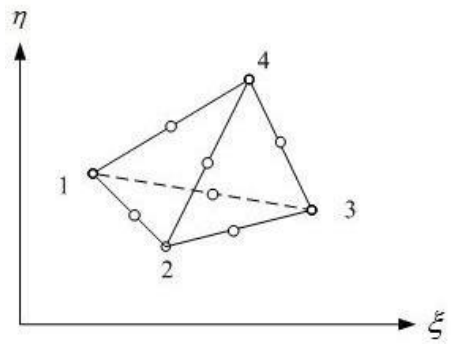


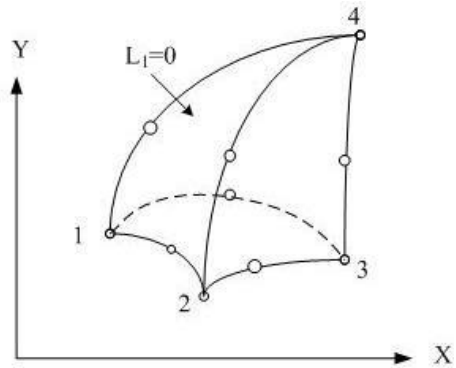

η
4
7
3
p(ξ,η)
8
ξ
6
1
5
2

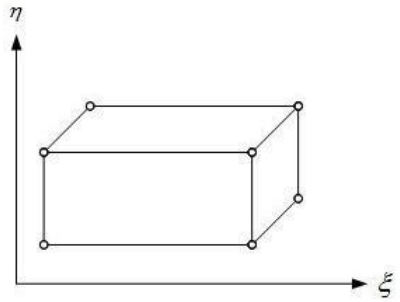

Y
7
3
4
P(x,y)
6
8
2
1
5
X

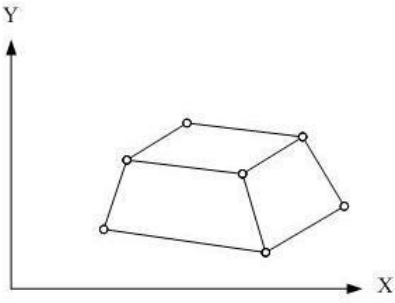

η
4
10
9
3
11
p(ξ,η)
8
ξ
12
7
1
5
6
2


Y
3
10
9
8
4
P(x,y)
11
7
12
5
2
6
1
X


η
4
1
3
2
ξ


Y
4
L1=0
1
3
2
X


η
ξ


Y
X

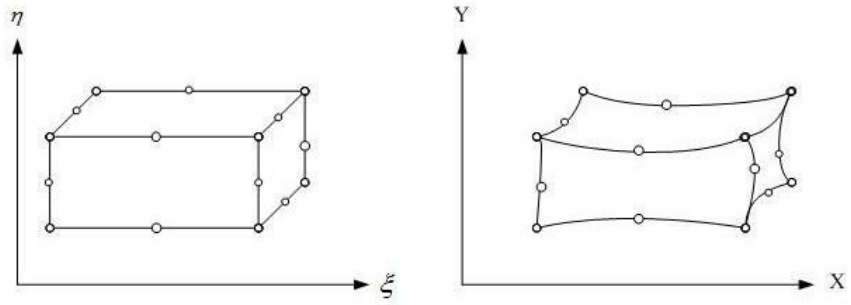


Fig 12 Mapping of isoparametric elements in global coordinate system

Coordinate Transformation

The geometry of an element may be expressed in terms of the interpolation functions as follows.

$$x = N_1x_1 + N_2x_2 + \ldots + N_nx_n = \sum_{i=1}^{n} N_ix_i$$

$$y = N_1y_1 + N_2y_2 + \ldots + N_ny_n = \sum_{i=1}^{n} N_iy_i$$

$$z = N_1z_1 + N_2z_2 + \ldots + N_nz_n = \sum_{i=1}^{n} N_iz_i$$

Where,

n=No. of Nodes

N =Interpolation Functions

x ,y ,z =Coordinates of Nodal Points of the Element

One can also express the field variable variation in the element as

$$\phi(\xi,\eta,\zeta) = \sum_{i=1}^{n} N_i(\xi,\eta,\zeta)\phi_i$$

As the same shape functions are used for both the field variable and description of element geometry, the method is known as isoparametric mapping. The element defined by such a method is known as an isoparametric element. This method can be used to transform the natural coordinates of a point to the Cartesian coordinate system and vice versa.

Numerical integration

Numerical integrations using Gauss Quadrature method can be extended to two and three dimensional cases in a similar fashion. Such integrations are necessary to perform for the analysis of plane stress/strain problem, plate and shell structures and for the three dimensional stress analysis.

Gauss Quadrature for Two-Dimensional Integrals

For two dimensional integration problems the above mentioned method can be extended by first evaluating the inner integral, keeping η constant, and then evaluating the outer integral. Thus,

$$I = \int_{-1}^{1}\int_{-1}^{1} \phi(\xi,\eta)\,d\xi\,d\eta \approx \int_{-1}^{1}\left[\sum_{i=1}^{n} w_i \phi(\xi_i,\eta)\right]d\eta \approx \sum_{i=1}^{n} w_j \left[\sum_{i=1}^{n} w_i \phi(\xi_i,\eta_j)\right]$$

Or,

$$I \approx \sum_{i=1}^{n}\sum_{j=1}^{n} w_i w_j \phi(\xi_i,\eta_j)$$

In a matrix form we can rewrite the above expression as

$$I \approx \begin{bmatrix} w_1 & w_2 & \cdots & w_n \end{bmatrix} \begin{bmatrix} \phi(\xi_1,\eta_1) & \phi(\xi_1,\eta_2) & & \phi(\xi_1,\eta_n) \\ \phi(\xi_2,\eta_1) & \phi(\xi_2,\eta_2) & & \phi(\xi_2,\eta_n) \\ & & \ddots & \\ \phi(\xi_n,\eta_1) & \phi(\xi_n,\eta_2) & & \phi(\xi_n,\eta_n) \end{bmatrix} \begin{Bmatrix} w_1 \\ w_2 \\ \vdots \\ w_n \end{Bmatrix}$$

Brick Elements

Various orders of elements of the parallelepiped family are shown in Fig. 13 shows the eight-node brick element with reference to a global Cartesian coordinate system and then with reference to natural coordinate system. The natural coordinates for the brick element can be relate Cartesian coordinate system by

$$\xi = \frac{x - \bar{x}}{a}, \qquad \eta = \frac{y - \bar{y}}{b} \quad and \quad \zeta = \frac{z - \bar{z}}{c}$$

Here,2*a*, 2*b* and 2*c*are the length, height and width of the element. The coordinate of the center of the brick element can be written as follows:

$$\bar{x} = \frac{x_1 + x_2}{2}, \qquad \bar{y} = \frac{y_1 + y_4}{2} \quad and \quad \bar{z} = \frac{z_1 + z_5}{2}$$

The nodal values in natural coordinate systems can be derived which is shown in Fig. 13 (b). With the above relations variations of *x*, *h* &*z* will be from -1 to +1. Now the interpolation function can be derived in several procedures as done in case of two dimensional rectangular elements. For example, the interpolation function can be derived by inspection in terms of natural coordinate system as follows:

$$N_i(\xi, \eta, \zeta) = \frac{1}{8}(1 + \xi\xi_i)(1 + \eta\eta_i)(1 + \zeta\zeta_i)$$

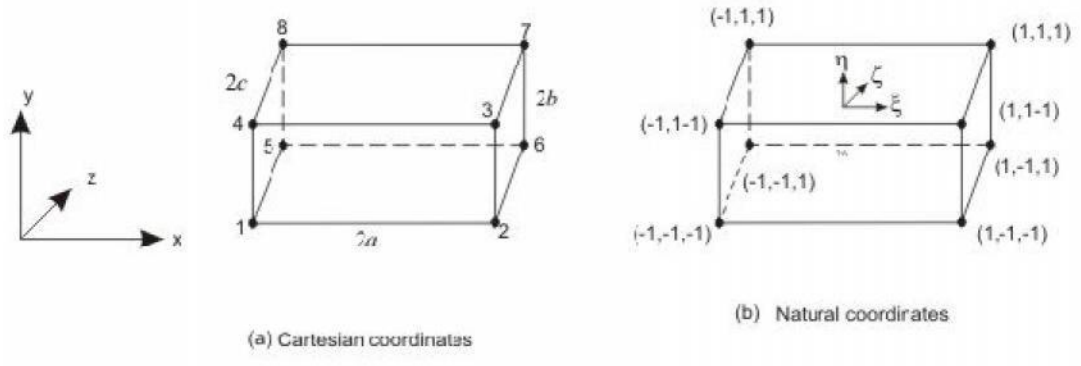


Fig. 13 Eight node brick element

By using field variable the following terms of the polynomial may be used for deriving the shape function for eight-node brick element.

$$\phi(\xi, \eta, \zeta) = \alpha_0 + \alpha_1\xi + \alpha_2\eta + \alpha_3\zeta + \alpha_4\xi\eta + \alpha_5\eta\zeta + \alpha_6\zeta\xi + \alpha_7\zeta\eta\xi$$

The above equation is incomplete but symmetric. However, such representations are quite often used and solution convergence is achieved in the finite element analysis. Again, the shape functions for three dimensional 8-node or 27-node brick elements can be derived using Lagrange interpolation function. For this we need to introduce interpolation function in the ζ-direction. Thus, for example, the Lagrange interpolation function for a three dimensional 8 node brick element can be obtained from the product of appropriate interpolation functions in the ξ, η and ζ directions. Therefore, the shape function will become

$$N_i(\xi,\eta,\zeta) = f_i(\xi) f_i(\eta) f_i(\zeta) \text{ Where, } i = 1,2,3, \ldots, \text{n-node}$$

Thus using the Lagrange interpolation function the shape function at node 1 can be expressed as

$$N_1(\xi,\eta,\zeta) = f_1(\xi) f_1(\eta) f_1(\zeta) = \frac{(\xi-\xi_2)}{(\xi_1-\xi_2)} \frac{(\eta-\eta_2)}{(\eta_1-\eta_2)} \frac{(\zeta-\zeta_2)}{(\zeta_1-\zeta_2)}$$

$$= \frac{(\xi-1)}{-1-(1)} \times \frac{(\eta-1)}{-1-(1)} \times \frac{(\zeta-1)}{-1-(1)} = \frac{1}{4}(1-\xi)(1-\eta)(1-\zeta)$$

Using any of the above concepts, the interpolation function for 8-node brick element can be found as follows:

$$N_1 = \frac{1}{4}(1-\xi)(1-\eta)(1-\zeta), \quad N_2 = \frac{1}{4}(1+\xi)(1-\eta)(1-\zeta),$$

$$N_3 = \frac{1}{4}(1+\xi)(1+\eta)(1-\zeta), \quad N_4 = \frac{1}{4}(1-\xi)(1+\eta)(1-\zeta),$$

$$N_5 = \frac{1}{4}(1-\xi)(1-\eta)(1+\zeta), \quad N_6 = \frac{1}{4}(1+\xi)(1-\eta)(1+\zeta),$$

$$N_7 = \frac{1}{4}(1+\xi)(1+\eta)(1+\zeta), \quad N_8 = \frac{1}{4}(1-\xi)(1+\eta)(1+\zeta)$$

The shape functions of rectangular parallelepiped elements with higher nodes can be derived in similar manner satisfying all necessary criteria.

Shell element

A shell is a curved surface, which by virtue of their shape can withstand both membrane and bending forces. A shell structure can take higher loads if, membrane stresses are predominant, which is primarily caused due to in-plane forces (plane stress condition). However, localized bending stresses will appear near load concentrations or geometric discontinuities. The shells are analogous to cable or arch structure depending on whether the shell resists tensile or, compressive stresses respectively. Few advantages using shell elements are given below.

1. Higher load carrying capacity
2. Lesser thickness and hence lesser dead load
3. Lesser support requirement
4. Larger useful space
5. Higher aesthetic value.

The example of shell structures includes large-span roof, cooling towers, piping system, pressure vessel, aircraft fuselage, rockets, water tank, arch dams, and many more. Even in the field of biomechanics, shell elements are used for analysis of skull, Crustaceans shape, red blood cells, etc.

Classification of Shells

Shell may be classified with several alternatives. Depending upon deflection in transverse direction due to transverse shear force per unit length, the shell can be classified into structurally thin or thick shell. Further, depending upon the thickness of the shell in comparison to the radii of curvature of the mid surface, the shell is referred to as geometrically thin or thick shell. Typically, if thickness to radii of curvature is less than 0.05, then the shell can be assumed as a thin shell. For most of the engineering application the thickness of shell remains within 0.001 to 0.05 and treated as thin shell.

Assumptions for Thin Shell Theory

Thin shell theories are basically based on Love-Kirchoff assumptions as follows.

1. As the shell deforms, the normal to the un-deformed middle surface remain straight and normal to the deformed middle surface undergo no extension. i.e., all strain components in the direction of the normal to the middle surface is zero.
2. The transverse normal stress is neglected.

Thus, above assumptions reduce the three dimensional problems into two dimensional.

Finite Element Formulation of a Degenerated Shell

Let consider a degenerated shell element, obtained by degenerating 3D solid element. The degenerated shell element as shown in Fig 14 has eight nodes, for which the analysis is carried out. Let $\xi, \eta)$ are the natural coordinates in the mid surface. And ς is the natural coordinate along thickness direction. The shape functions of a two dimensional eight node isoparametric element are:

$$N_1 = \frac{(1-\xi)(1-\eta)(-\xi-\eta-1)}{4} \qquad N_5 = \frac{(1+\xi)(1-\xi)(1-\eta)}{2}$$

$$N_2 = \frac{(1+\xi)(1-\eta)(\xi-\eta-1)}{4} \qquad N_6 = \frac{(1+\xi)(1+\eta)(1-\eta)}{2}$$

$$N_3 = \frac{(1+\xi)(1+\eta)(\xi+\eta-1)}{4} \qquad N_7 = \frac{(1+\xi)(1-\xi)(1+\eta)}{2}$$

$$N_4 = \frac{(1-\xi)(1+\eta)(-\xi+\eta-1)}{4} \qquad N_8 = \frac{(1-\xi)(1+\eta)(1-\eta)}{2}$$

The position of any point inside the shell element can be written in terms of nodal coordinates as

$$\begin{Bmatrix} x \\ y \\ z \end{Bmatrix} = \sum_{i=1}^{8} N_i(\xi,\eta) \left\{ \frac{1+\varsigma}{2} \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix}_{top} + \frac{1-\varsigma}{2} \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix}_{bottom} \right\}$$

Since, ς is assumed to be normal to the mid surface, the above expression can be rewritten in terms of a vector connecting the upper and lower points of shell as

$$\begin{Bmatrix} x \\ y \\ z \end{Bmatrix} = \sum_{i-1}^{8} N_i(\xi,\eta) \left\{ \frac{1}{2} \left\{ \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix}_{top} + \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix}_{bottom} \right\} + \frac{\varsigma}{2} \left\{ \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix}_{top} - \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix}_{bottom} \right\} \right\}$$

$$\begin{Bmatrix} x \\ y \\ z \end{Bmatrix} = \sum_{i=1}^{8} N_i(\xi,\eta) \left\{ \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix} + \frac{\varsigma}{2} V_{3i} \right\}$$

Where,

$$\begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix} = \frac{1}{2} \left\{ \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix}_{top} + \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix}_{bottom} \right\} \text{ and, } V_{3i} = \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix}_{top} - \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix}_{bottom}$$

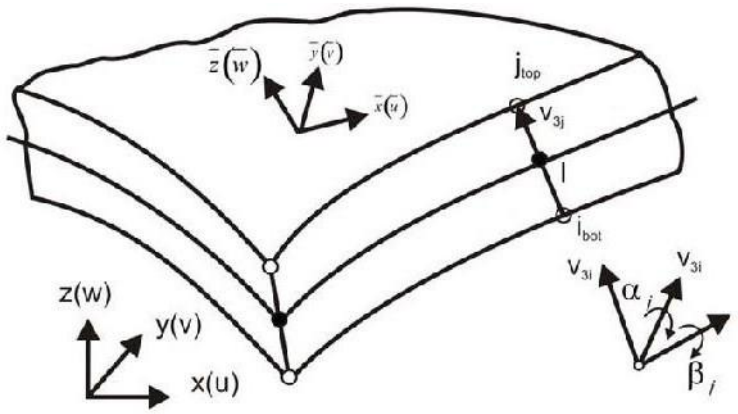


Fig 14 Local and global coordinates

For small thickness, the vector ***V3i*** can be represented as a unit vector *tiv3i*:

$$\begin{Bmatrix} x \\ y \\ z \end{Bmatrix} = \sum_{i=1}^{8} N_i(\xi,\eta) \left\{ \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix} + \frac{\varsigma}{2} t_i v_{3i} \right\}$$

Where, *ti* is the thickness of shell at *ith* node. In a similar way, the displacement at any point ofthe shell element can be expressed in terms of three displacements and two rotation components about two orthogonal directions normal to nodal load vector ***V3i*** as,

$$\begin{Bmatrix} u \\ v \\ w \end{Bmatrix} = \sum_{i=1}^{8} N_i(\xi,\eta) \left\{ \begin{Bmatrix} u_i \\ v_i \\ w_i \end{Bmatrix} + \frac{\varsigma t_i}{2} \begin{bmatrix} v_{1i} & -v_{2i} \end{bmatrix} \begin{Bmatrix} \alpha_i \\ \beta_i \end{Bmatrix} \right\}$$

Where, σi, β_i ae the rotations of two unit vectors v1i & v2i about two orthogonal directions normal to nodal load vector V3i.The values of v1i and v2i can be calculated in following way:

The coordinate vector of the point to which a normal direction is to be constructed may be defined as

$$x = x\hat{i} + y\hat{j} + z\hat{k}$$

In which, *i*ˆ, ˆ*j*, *k*ˆ are three (orthogonal) base vectors. Then, 1*i V* is the cross product of ˆ*i* & *V3i* as shown below

$$V_{1i} = \hat{i} \times V_{3i} \quad \& \quad V_{2i} = V_{3i} \times V_{1i}$$

$$v_{1i} = V_{1i} \Big/ |V_{1i}| \quad \& \quad v_{2i} = V_{2i} \Big/ |V_{2i}|$$

Jacobian matrix

The Jacobian matrix for eight node shell element can be expressed as,

$$[J]=\begin{bmatrix} \sum_{i=1}^{8}\left(x_i+tx_i^*\right)\frac{\partial N_i}{\partial \xi} & \sum_{i=1}^{8}\left(y_i+ty_i^*\right)\frac{\partial N_i}{\partial \xi} & \sum_{i=1}^{8}\left(z_i+tz_i^*\right)\frac{\partial N_i}{\partial \xi} \\ \sum_{i=1}^{8}\left(x_i+tx_i^*\right)\frac{\partial N_i}{\partial \eta} & \sum_{i=1}^{8}\left(y_i+ty_i^*\right)\frac{\partial N_i}{\partial \eta} & \sum_{i=1}^{8}\left(z_i+tz_i^*\right)\frac{\partial N_i}{\partial \eta} \\ \sum_{i=1}^{8}Nx_i^* & \sum_{i=1}^{8}Ny_i^* & \sum_{i=1}^{8}Nz_i^* \end{bmatrix}$$

Strain displacement matrix

The relationship between strain and displacement is described by

$\{e\}=[B]\{d\}$

Where, the displacement vector will become:

$\{ d\}^{t} = \{u_1\ v_1\ w_1\ v_{11} v_{21}\ .....u_8\ v_8\ w_8\ \}$

And the strain components will be

$$[\varepsilon]=\begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial u}{\partial y}+\frac{\partial v}{\partial x} \\ \frac{\partial v}{\partial z}+\frac{\partial w}{\partial y} \\ \frac{\partial w}{\partial x}+\frac{\partial u}{\partial z} \end{Bmatrix}$$

$$\begin{bmatrix} \frac{\partial u}{\partial \xi} & \frac{\partial v}{\partial \xi} & \frac{\partial w}{\partial \xi} \\ \frac{\partial u}{\partial \eta} & \frac{\partial v}{\partial \eta} & \frac{\partial w}{\partial \eta} \\ \frac{\partial u}{\partial \varsigma} & \frac{\partial v}{\partial \varsigma} & \frac{\partial w}{\partial \varsigma} \end{bmatrix} = \sum_{i=1}^{8} \begin{Bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \\ 0 \end{Bmatrix} \begin{bmatrix} u_i & v_i & w_i \end{bmatrix} - \sum_{i=1}^{8} \frac{t_i v_{2i}}{2} \begin{Bmatrix} \varsigma \frac{\partial N_i}{\partial \xi} \\ \varsigma \frac{\partial N_i}{\partial \eta} \\ N_i \end{Bmatrix} \times \begin{bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \end{bmatrix}_i^T + \sum_{i=1}^{8} \frac{t_i v_{1i}}{2} \begin{Bmatrix} \varsigma \frac{\partial N_i}{\partial \xi} \\ \varsigma \frac{\partial N_i}{\partial \eta} \\ N_i \end{Bmatrix} \times \begin{bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \end{bmatrix}_i^T$$

Stress strain relation

The stress strain relationship is given by

$\{\sigma\}=[D]\{\varepsilon\}$

$\{\sigma\}=[D][B]\{d\}$

Where, the stress strain relationship matrix is represented by

$$[D] = \frac{E}{1-\mu^2} \begin{bmatrix} 1 & \mu & 0 & 0 & 0 \\ \mu & 1 & 0 & 0 & 0 \\ 0 & 0 & \frac{1-\mu}{2} & 0 & 0 \\ 0 & 0 & 0 & \frac{\alpha(1-\mu)}{2} & 0 \\ 0 & 0 & 0 & 0 & \frac{\alpha(1-\mu)}{2} \end{bmatrix}$$

The value of shear correction factor *a* is considered generally as 5/6. The above constitutive matrix can be split into two parts ([*Db*] and [*Ds*])for adoption of different numerical integration schemes for bending and shear contributions to the stiffness matrix.

$$[D] - \begin{bmatrix} [D_b] & \vdots & [0] \\ \cdots & \cdots & \cdots \\ [0] & \vdots & [D_s] \end{bmatrix}$$

Thus,

$$[D_b] = \frac{E}{1-\mu^2} \begin{bmatrix} 1 & \mu & 0 \\ \mu & 1 & 0 \\ 0 & 0 & \frac{1-\mu}{2} \end{bmatrix}$$

and

$$[D_s] = \frac{E\alpha}{2(1+\mu)} \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}$$

Element stiffness matrix

Finally, the stiffness matrix for the shell element can be computed from the expression

$$[k]= \iiint [B]^T [D][B]d\Omega$$

However, it is convenient to divide the elemental stiffness matrix into two parts: (i) bending and membrane effect and (ii) transverse shear effects. This will facilitate the use of appropriate order of numerical integration of each part. Thus,

$$[k]= [k]_b [k]_s$$

Where, contribution due to bending and membrane effects to stiffness is denoted as $[k]b$ and transverse shear contribution to stiffness is denoted as $[k]s$ and expressed in the following form.

$$[k]_b = \iiint [B]_b^T [D]_b [B]_b \, d\Omega \text{ and } [k]_s = \iiint [B]_s^T [D]_s [B]_s \, d\Omega$$

Numerical procedure will be used to evaluate the stiffness matrix. A 2 ×2 Gauss Quadrature can be used to evaluate the integral of $[k]b$ and one point Gauss Quadrature may be used to integrate $[k]s$ to avoid shear locking effect.

Plate Bending Elements

The elastic stability analysis of rectangular plates is discussed in this section. The total potential energy for plate are expressed as

$$\pi = \frac{D}{2}\int_{-a}^{a}\int_{-b}^{b}\left\{(\nabla^2 w)^2 + 2(1-\mu)\left[\left(\frac{\partial^2 w}{\partial x \partial y}\right)^2 - \left(\frac{\partial^2 w}{\partial x^2}\right)\left(\frac{\partial^2 w}{\partial y^2}\right)\right]\right\}dxdy$$
$$-\frac{1}{2}\int_{-a}^{a}\int_{-b}^{b}\left\{F_x\left(\frac{\partial w}{\partial x}\right)^2 + 2F_{xy}\frac{\partial w}{\partial y}\frac{\partial w}{\partial x} + F_y\left(\frac{\partial w}{\partial y}\right)^2\right\}dxdy$$

Here, *Fx*,*Fy*,and *Fxy*, are the in-plane edge load and compressive load is considered as positive. For, finite element formulation the deflection in aboveexpression needs to convert in terms of nodal displacements in the element. to the following form using interpolation functions.

$$\pi = \frac{1}{2}\{d\}^T[k_F]\{d\} - \frac{F_x}{2}\{d\}^T[k_{Gx}]\{d\} - \frac{F_y}{2}\{d\}^T[kK_{Gy}]\{d\} - \frac{F_{xy}}{2}\{d\}^T[k_{Gxy}]\{d\}$$

$$k_{Gx} = \iint [N'_x]\{N'_x\}dxdy$$

$$k_{Gy} = \iint [N'_y]\{N'_y\}dxdy$$

$$k_{Gxy} = \iint [N'_x]\{N'_y\}dxdy$$

Where [N'$_x$] and [N'$_y$] indicate partial derivative of ??? with respect to *x* and *y* respectively. Thus, the equation of buckling becomes

$$[k_F]\{d\} - F_x[k_{Gx}]\{d\} - F_y[k_{Gy}]\{d\} - F_{xy}[k_{Gxy}]\{d\} = 0$$

If the in-plane loads have a constant ratio to each other at all time during their buildup, the above equation can be expressed as follows

$$[k_F]\{d\} = P^*(\alpha[k_{Gx}] + \beta[k_{Gy}] + \gamma[k_{Gxy}])\{d\}$$

The term P^* is called the load factor, and α, β and γ are constants relating the in-plane loads in the plate member. Solving the above expression, the buckling mode shapes are possible to determine.

SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF BUILDING AND ENVIRONMENT

DEPARTMENT OF CIVIL ENGINEERING

UNIT – III - MESHING AND SOLUTION PROBLEMS – SCIA5201

UNIT-III MESHING AND SOLUTION PROBLEMS

The stiffness matrix of the basic element is contained in the new matrix as a submatrix. The stiffness matrices of higher order elements are built by a similar process if a higher order element is coded into the finite element program, it includes stiffness matrices of all lower order elements. In the process of refinement if a higher order element is chosen, the previously computed stiffness coefficients would still be valid. Hence, only a few additional coefficients have to be evaluated. The method is easier than the conventional p - method of increasing the polynomial order where the computation of the entire higher order element stiffness matrix is required.

Refinement using hierarchical elements is a-posteriori and appears to be attractive. However more research work needs to be done in this area.

Mesh refinement

The user needs to select the number of nodes and elements in the model. The selection may be the one that leads to the best description of the domain geometrically. For example, a curved surface could be modeled **by** a series of interconnected flat rectangular facets. The larger the number of facets, the better is the model. The selection may also be based upon intuition, past experience and engineering judgment. The mesh obtained may be adequate in some cases. In other cases, especially when singularities are present, the mesh may not be adequate to obtain the results to the accuracy desired. In such cases, the meshes need to be refined.

REFINEMENT PROCESS

There are three ways of refining a finite element mesh:

a) The H-method: This method increases the number of elements and hence decreases the element size while keeping the polynomial order of the shape function constant.

b) The P-method: This method increases the polynomial order of the interpolation function while keeping the number of elements in the model constant.

c) The R-method: This method redistributes the nodes while keeping the element number and the polynomial order of the interpolation function constant.

H - Method

This method is primarily based on the choice of characteristic length of the elements. "Characteristic length " is referred to in a generalized sense and is required to define the element topologically. A linear element requires one characteristic length, whereas an element of rectangular shape requires two characteristic lengths and a triangular element requires three characteristic lengths for its definition. In the triangular element the three length information's may be any combination of lengths and angles.

Instead of expressing the functional in terms of the position vectors of the nodes it can be expressed as a function of the element characteristic lengths as

$$\pi = \pi\ (u_i, h_{ik}) \qquad \text{(a)}$$

where, h_{ik} is the element characteristic length, 1 is the index on the Characteristic length for element k

Also, note that there will be geometrical constraints on hk**.** For example, the sum of the element lengths in a particular direction should be equal to the overall dimension of the model in that direction.

$$\frac{\partial \pi}{\partial h} = \frac{\partial K}{\partial h} u = 0 \qquad \text{(b)}$$

Solving equation (b) along with the constraints yield the characteristic lengths and hence defines the best mesh. Equation (b) is equivalent to cast in the frame work of characteristic lengths. Therefore the solution as indicated is difficult. A practical procedure using this method consists of selecting a coarse initial mesh, solving the equilibrium equations and computing the residue rk on each element. The set of elements with large values of residues is the region that needs to be refined. The identified region can be refined by sub-dividing the elements, thus creating new regions, or by deleting all the elements in the region and replacing them **by** finer elements. However, the new elements need to be of the same type as those in the initial mesh. The equations of the new model are solved and the residues are computed. If the values of the residues are still large, the refinement procedure can be repeated. Indeed , it could be used iteratively until the solution meets the prescribed accuracy.

P - Method

This method is primarily based on the choice of the order of the interpolation function, which in practice, translates to the choice of element type. For example, in a two dimensional domain, the basic triangular element with three nodes at the three vertices uses a linear shape function (p-l). In order to choose quadratic shape functions (p=2), the triangular element with six nodes, three at the vertices and three at mid-side locations, has to be selected. Similarly, for cubic functions, an element with nine nodes is selected.

Higher order elements generally provide better description of the domain geometrically. They are particularly useful in regions where the use of lower order elements would result in a mesh with poor aspect ratios in those elements. From the point of view of solution accuracy, higher order elements are usually more accurate than the lower order elements. But this does not mean that increasing the polynomial order indiscriminately will always provide point-wise convergence to the exact solution. The argument is based on the theory of interpolation. Prenter states that this notion on convergence was first dispelled by Meray and later by Range. He illustrates this with the function f(x) = 1/(1+5x2) being interpolated by Lagrange polynomial of order **5** and **15** with evenly spaced odes in the interval [-1,1] which display divergence at - 1 and 1. Although the example is for a continuous interpolation function rather than a piecewise function, as in a finite element model, it shows that there is good reason to exercise caution in increasing the polynomial order.

A - priori and a - Posteriori methods

The classification of methods into a-priori and a-posteriori refers to refinement before and after the solution of the equilibrium equations. In a finite element program the solution process is one that needs much of computer time. If discretization errors can be estimated a-priori, then the mesh can be suitably altered to obtain the best accuracy possible by solving the equations only once. Unfortunately there are no practical priory methods available. The author has not found any in the literature survey. This study is an attempt to provide one. There are several aposteriori methods available for refinement.

SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF BUILDING AND ENVIRONMENT

DEPARTMENT OF CIVIL ENGINEERING

UNIT – IV -NONLINEAR, VIBRATION AND THERMAL PROBLEMS– SCIA5201

UNIT IV
NONLINEAR, VIBRATION AND THERMAL PROBLEMS

Nonlinear problems

Various non-linear problems in finite element analysis may be group into the following three categories.

1. Material non-linear problems

2. Geometric non-linear problems

3. Non-linear boundary or initial conditions

Nonlinear Material Behavior

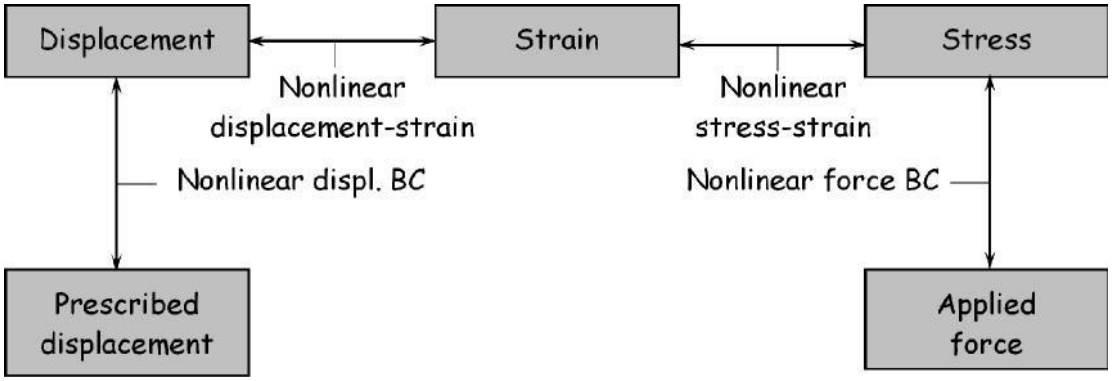
This is one of the most common forms of nonlinearity, and would include nonlinear elastic, plastic, and visco elastic behavior. For thermal problems, a temperature dependent thermal conductivity will produce nonlinear equations.

Large Deformation Theory (Geometric Nonlinearity)

If a continuum body under study undergoes large finite deformations, the strain-displacement relations will become nonlinear. Also for structural mechanics problems under large deformations, the stiffness will change with deformation thus making the problem nonlinear. Buckling problems are also nonlinear.

Nonlinear Boundary or Initial Conditions

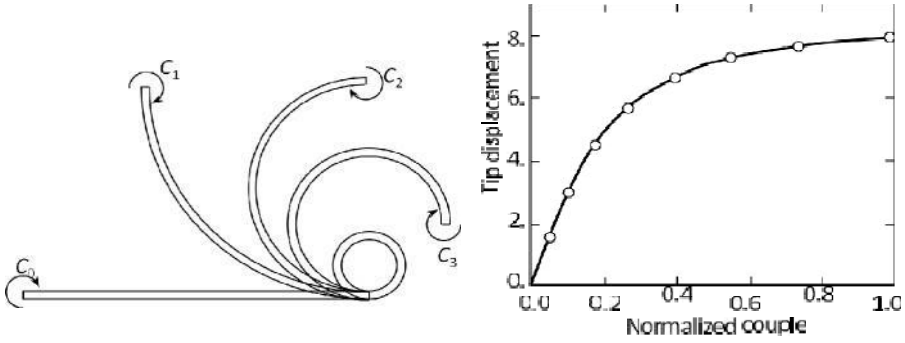
Problems involving contact mechanics normally include a boundary condition that depends on the deformation thereby producing a nonlinear formulation. Thermal problems involving melting or freezing (phase change) also include such nonlinear boundary conditions.

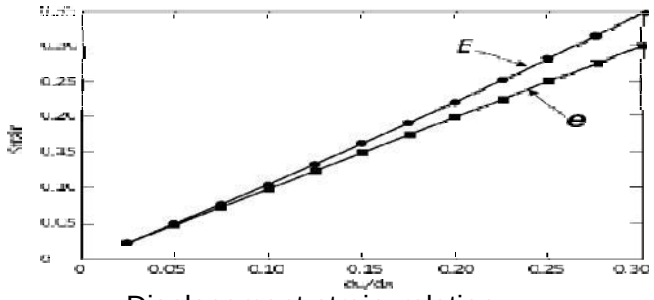


Non linearity in structural problem

GEOMETRIC NONLINEARITY

Relations among kinematic quantities (i.e., displacement, rotation and strains) are nonlinear





Displacement-strain relation

i. E has a higher-order term

ii. (du/dx) << 1 → e(x) ~ E(x).

Domain of integration

i. Undeformed domain W_0

ii. Deformed domain W_X

$$a(\mathbf{u}, \bar{\mathbf{u}}) = \iint_{\Omega} \varepsilon(\bar{\mathbf{u}}) : \sigma(\mathbf{u})\, d\Omega$$

MATERIAL NONLINEARITY

Linear (elastic) material

$$\{ \sigma \} = [D]\{ \varepsilon \}$$

Only for infinitesimal deformation

Nonlinear (elastic) material

i. [C] is not a constant but depends on deformation

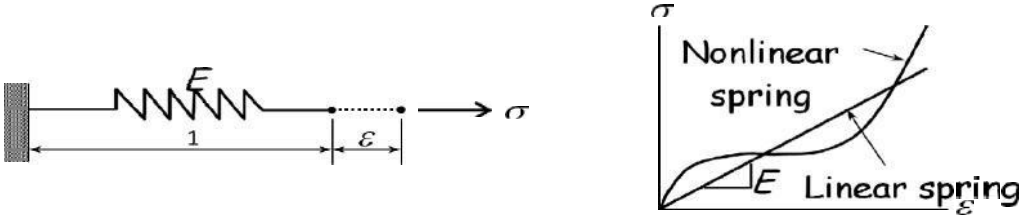
ii. Stress by differentiating strain energy density U

iii. Linear material:

$$U = \frac{1}{2} E\varepsilon^2$$

$$\sigma = \frac{dU}{d\varepsilon} = E\varepsilon$$

Stress is a function of strain (deformation): potential, path independent



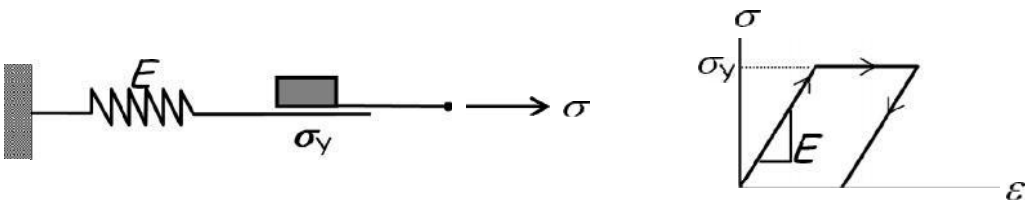
Linear and nonlinear elastic spring models

Elasto-plastic material (energy dissipation occurs)

i. Friction plate only support stress up to sy

ii. Stress cannot be determined from stress alone

iii. History of loading path is required: path-dependent

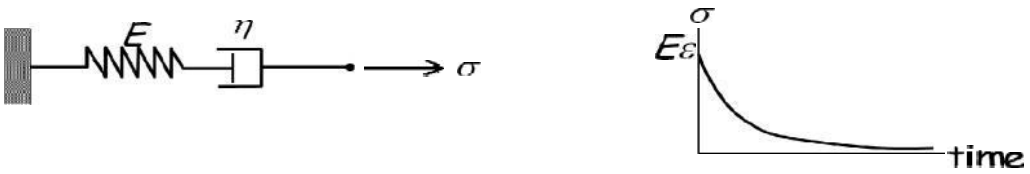


Elasto-plastic spring model

Visco-elastic material

i. Time-dependent behavior

ii. Creep, relaxation

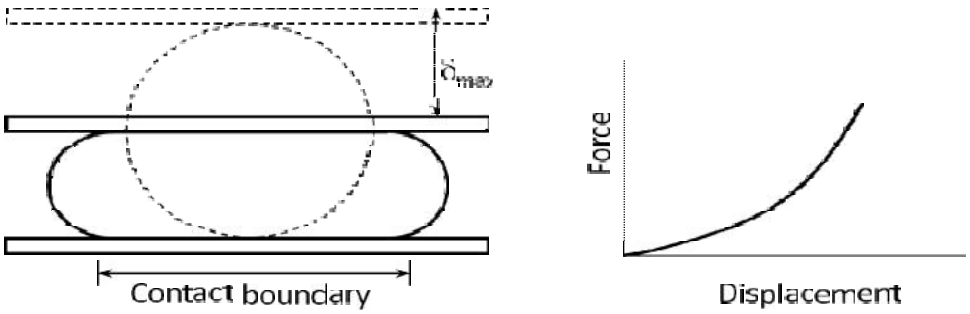


Visco-elastic spring model

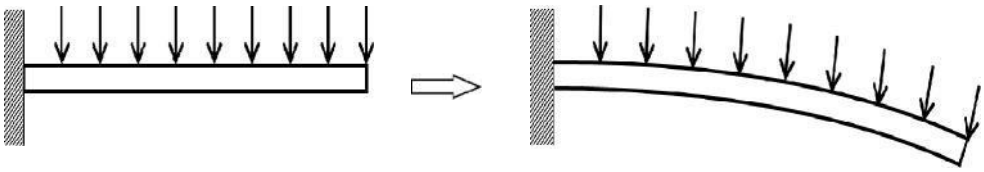
Boundary and Force Nonlinearities

Nonlinear displacement BC (kinematic nonlinearity)

Contact problems, displacement dependent conditions

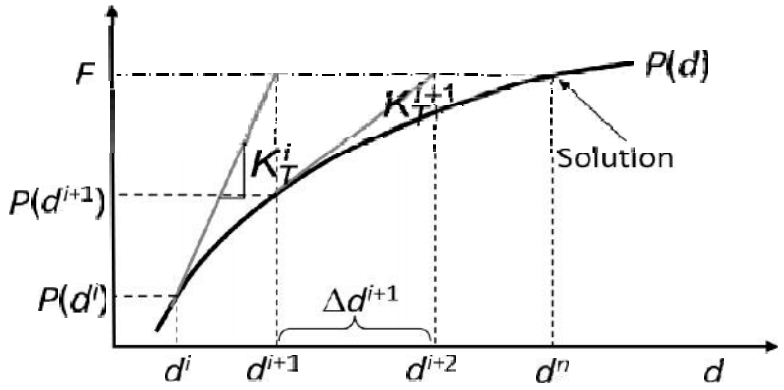


Nonlinear force BC (Kinetic nonlinearity)



NON LINEAR ANALYSIS

Newton-Raphson Method



i. Most popular method

ii. Assume $\mathbf{d}^i$ at i-th iteration is known

iii. Looking for $\mathbf{d}^{i+1}$ from first-order Taylor series expansion

$$\mathbf{P}(\mathbf{d}^{i+1}) \approx \mathbf{P}(\mathbf{d}^i) + \mathbf{K}_T^i(\mathbf{d}^i) \cdot \Delta\mathbf{d}^i = \mathbf{F}$$

$$\mathbf{K}_T^i(\mathbf{d}^i) \equiv \left(\frac{\partial \mathbf{P}}{\partial \mathbf{d}}\right)_i$$

: **Jacobian matrix or Tangent stiffness matrix**

iv. Solve for incremental solution

$$\mathbf{K}_T^i \Delta\mathbf{d}^i = \mathbf{F} - \mathbf{P}(\mathbf{d}^i)$$

v. Update solution

$$\mathbf{d}^{i+1} = \mathbf{d}^i + \Delta\mathbf{d}^i$$

Consistent System Matrices

To do dynamic and vibration finite element analysis, you need at least a mass matrix to pair with the stiffness matrix. As a general rule, the construction of the master mass matrix M largely parallels of the master stiffness matrix K. Mass matrices for individual elements are formed in local coordinates, transformed to global, and merged into the master mass matrix following exactly the same techniques used for K. In practical terms, the assemblers for K and M can be made identical. This procedural uniformity is one of the great assets of the Direct Stiffness Method. A notable difference with the stiffness matrix is the possibility of using a *diagonal* mass matrix based on direct lumping. A master diagonal mass matrix can be stored simply as a vector. If all entries are nonnegative, it is easily inverted, since the inverse of a diagonal matrix is also diagonal. Obviously a lumped mass matrix entails significant computational advantages for calculations that involve M−1. This is balanced by some negative aspects that are examined in some detail later.

Mass Matrix Construction

The master mass matrix is built up from element contributions, and we start at that level. The construction of the mass matrix of individual elements can be carried out through several methods. can be categorized into three groups: direct mass lumping, variational mass lumping, and template mass lumping. The last group is more general in that includes all others. Variants of the first two techniques are by now standard in the FEM literature, and implemented in all general purpose codes.

Direct Mass Lumping

The total mass of element *e* is directly apportioned to nodal freedoms, ignoring any cross coupling. The goal is to build a *diagonally lumped mass matrix* or DLMM, denoted here by M*eL* .As the simplest example, consider a 2-node prismatic bar element with length *L*, cross section area *A*, and mass density ρ, which can only move in the axial direction *x*,. The total mass of the element is *Me* = ρ *AL*. This is divided into two equal parts and assigned to each end

Dynamic condensation

The accuracy of the resulting reduced model is generally very low for dynamic problems. To achieve reasonably accurate results, the masters must be chosen with great care and number of masters should be greater than the number of modes interested. To alleviate the limitations, the inertia effects could be partially or fully included in the condensation. The corresponding condensation approaches are generally called dynamic condensation

The equation of motion is cast as a shifted Eigen problem. A shift value, f, is introduced into the set of equations describing the dynamic system, thus

$$[[K_n]-(\lambda-f)[M_n]]\{x_n\}=\{0\}$$

The terms are rearranged to group the constant term f times the mass matrix with the stiffness matrix to yield

$$[[[K_n]+f[M_n]]-\lambda[M_n]]\{x_n\}=\{0\}$$

Then let a new system matrix [D] be used to describe the 'effective' stiffness matrix as

$$[D_n]=[[K_n]+f[M_n]]$$

This 'effective' stiffness equation

$$[D_n]\{x_n\}=\{F_n\}$$

can be partitioned into the 'a' active DOF and the 'd' deleted or omitted DOF to form two equations given as

$$\begin{bmatrix}[D_{aa}] & [D_{ad}]\\ [D_{da}] & [D_{dd}]\end{bmatrix}\begin{Bmatrix}x_a\\ x_d\end{Bmatrix}=\begin{Bmatrix}F_a\\ F_d\end{Bmatrix}$$

Assuming that the forces on the deleted DOF are zero, then the second equation can be written as

$$[D_{da}]\{x_a\}+[D_{dd}]\{x_d\}=\{0\}$$

which can be solved for the displacement at the deleted DOF as

$$\{x_d\}=-[D_{dd}]^{-1}[D_{da}]\{x_a\}$$

The first equation can be written as

$$[D_{aa}]\{x_a\}+[D_{ad}]\{x_d\}=\{F_a\}$$

and upon substituting for the 'd' deleted DOF this equation becomes

$$[D_{aa}]\{x_a\}+[D_{ad}][D_{dd}]^{-1}[D_{da}]\{x_a\}=\{F_a\}$$

This can be manipulated to yield the desired transformation to be

$$[T_f]=\begin{bmatrix}[I]\\[t_f]\end{bmatrix}=\begin{bmatrix}[I]\\-[D_{dd}]^{-1}[D_{da}]\end{bmatrix}$$

Using this transformation, the reduced stiffness can be written as

$$\left[K_a^f\right]=[T_f]^T[K_n][T_f]$$

This same transformation can be applied to the mass matrix given by

$$\left[M_a^f\right]=[T_f]^T[M_n][T_f]$$

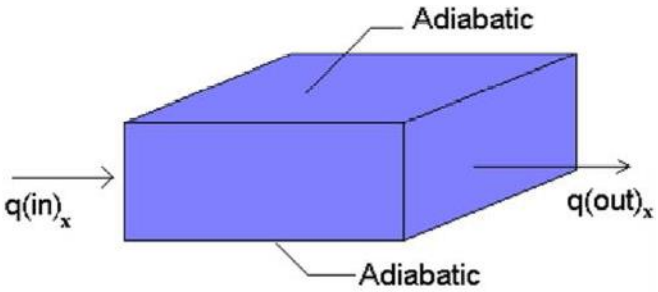
THERMAL ANALYSIS

One dimensional conduction

$$q_x=-K_{xx}\cdot\frac{dT}{dx}$$

K – thermal conductivity

dT = temperature change



$$E_{in}+E_{gen}=\Delta U+E_{out}$$

$$q_{in}\cdot A\cdot dt+Q\cdot A\cdot dx\cdot dt=\Delta U+q_{out}\cdot A\cdot dt$$

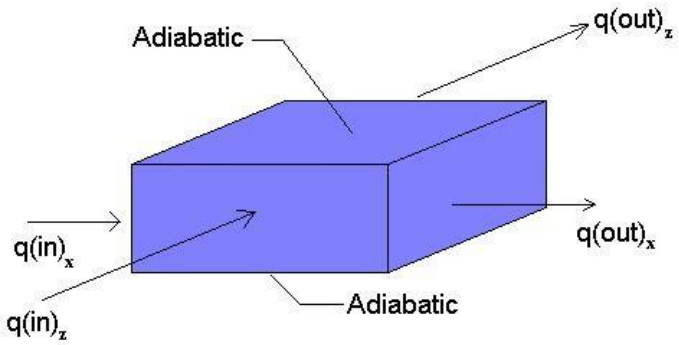
q = heat conducted E = kinetic energy

Q = internal heat source U = stored energy

A = cross-sectional area t = time

Two-Dimensional Conduction

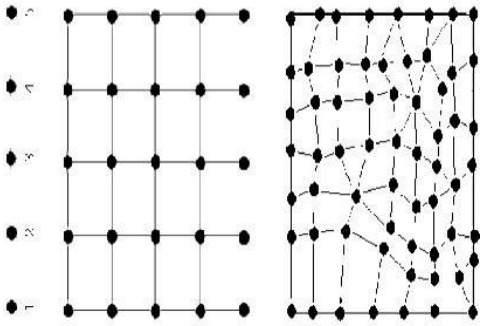
$$q_x = -K_{xx} \cdot \frac{dT}{dx} + -K_{zz} \cdot \frac{dT}{dz}$$



$$E_{in} + E_{gen} = \Delta U + E_{out}$$

$$q_{inX} \cdot A \cdot dt + q_{inZ} \cdot A \cdot dt + Q \cdot A \cdot dx \cdot dt = \Delta U + q_{outX} \cdot A \cdot dt + q_{outZ} \cdot A \cdot dt$$

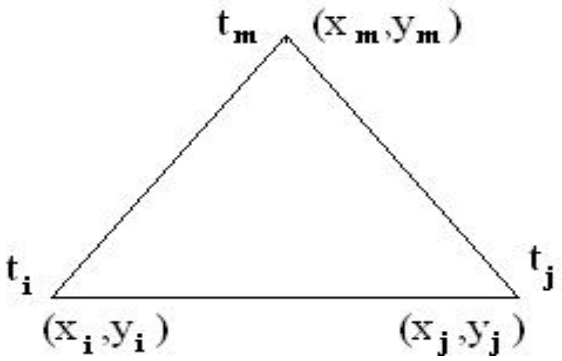
Finite Element 2-D Conduction
Select Element Type



i. 1-d elements are lines
ii. 2-d elements are either **triangles**, quadrilaterals, or a mixture as shown
iii. Label the nodes so that the difference between two nodes on any element is minimized.

Finite Element 2-D Conduction

1. Assume (Choose) a Temperature Function



Assume a linear temperature function for each element as:

$$t(x,y) = a_1 + a_2 x + a_3 y$$

$$\{\Psi\} = \{a_1 + a_2 x + a_3 y\} = [1 \text{ x y}] \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \end{Bmatrix}$$

Where u and v describe temperature gradients at (x_i, y_i).

2.Assume (Choose) a Temperature Function

$$T = N_i t_i + N_j t_j + N_m t_m$$

$$[T] = [N_i + N_j + N_m] \begin{Bmatrix} t_i \\ t_j \\ t_m \end{Bmatrix}$$

T = temperature function

N = shape function

t = nodal temperature

3.Define Temperature Gradient Relationships

$$\{g\} = \begin{Bmatrix} \frac{\partial T}{\partial x} \\ \frac{\partial T}{\partial y} \end{Bmatrix} = \begin{bmatrix} \frac{\partial N_i}{\partial x} & \frac{\partial N_j}{\partial x} & \frac{\partial N_m}{\partial x} \\ \frac{\partial N_i}{\partial y} & \frac{\partial N_j}{\partial y} & \frac{\partial N_m}{\partial y} \end{bmatrix} \begin{Bmatrix} t_i \\ t_j \\ t_m \end{Bmatrix}$$

$$[B] = \frac{\partial}{\partial x}[N] = \frac{1}{|x|} \begin{bmatrix} \beta_i & \beta_j & \beta_m \\ \gamma & \gamma & \gamma \end{bmatrix}$$

Analogous to strain matrix: {g}=[B]{t}

[B] is derivative of [N]

Heat flux/Temperature Gradient :

$$\begin{Bmatrix} q_x \\ q_y \end{Bmatrix} = - \begin{bmatrix} K_{xx} & 0 \\ 0 & K_{yy} \end{bmatrix} \{g\} = -[D]\{g\}$$

4. Derive Element Conduction Matrix and Equations

$$[k] = \underbrace{\iiint_V [B]^T [D][B] dV}_{\text{Conduction}} + \underbrace{\iint_S h[N]^T [N] dS}_{\text{Convection}}$$

$$= tA[B]^T [D][B] + hP \int_0^L \begin{Bmatrix} 1 - \frac{x}{L} \\ \frac{x}{L} \end{Bmatrix} \begin{bmatrix} 1 - \frac{x}{L} & \frac{x}{L} \end{bmatrix} dx$$

$$= \frac{AK_{xx}}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} + \frac{hPL}{6} \begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix}$$

5. Derive Element Conduction Matrix and Equations

$$\{f_Q\} = Q\iiint_V [V]^T dV = \frac{QV}{3}\begin{Bmatrix}1\\1\\1\end{Bmatrix} \text{ for constant heat source}$$

$$\{f\} = [k]\{t\}$$

for each element

Stiffness matrix is general term for a matrix of known coefficients being multiplied by unknown degrees of freedom, i.e., displacement OR temperature, etc. Thus, the element conduction matrix is often referred to as the stiffness matrix.

2. Assemble Element Equations, Apply BC's

$$\{F\} = [K]\{t\}$$

From here on virtually the same as structural approach. Heat flux boundary conditions already accounted for in derivation. Just substitute into above equation and solve for the following:

3. Solve for Nodal Temperatures

4. Solve for Element Temperature Gradient & Heat Flux

SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF BUILDING AND ENVIRONMENT

DEPARTMENT OF CIVIL ENGINEERING

UNIT – V -DYNAMIC ANALYSIS AND SOFTWARE APPLICATION– SCIA5201

UNIT V
DYNAMIC ANALYSIS AND SOFTWARE APPLICATION

Dynamic equation of motion

In dynamic problems the displacements, velocities, strains, stresses and loads are all time dependent. The procedure involved in deriving the FE equations of a dynamic problem can be stated by the following steps:

1. Idealize the body into E finite elements
2. Assume the displacement model of element e as

$$\vec{U}(x,y,z,t) = \begin{Bmatrix} u(x,y,z,t) \\ v(x,y,z,t) \\ w(x,y,z,t) \end{Bmatrix} = [N(x,y,z)]\vec{Q}^{(e)}(t)$$

3. Derive the element characteristic (stiffness and mass) matrices and characteristic (load) vector.

$$\vec{\varepsilon} = [B]\vec{q}^{(e)}$$

$$\vec{\sigma} = [D]\vec{\varepsilon} = [D][B]\vec{Q}^{(e)}$$

4. Assemble the element matrices and vectors and derive the overall system equations of motion.
5. Solve the equation of motion by applying the boundary conditions.

Consistent and lumped mass matrices

The above mass matrix is called as **"consistent"** mass matrix of the element. It is called consistent because the same displacement model that is used for deriving the element stiffness matrix is used to for the derivation of mass matrix.

It is interest to note that several dynamic problems have been and are being solved with simpler forms of mass matrices. The simplest form of mass matrix that can be used is that obtained by placing point (concentrated) mass mi at node point I in the directions of assumed displacement degrees of freedom.

The concentrated masses refer to translational and rotational inertia of the element and are calculated by assuming that the material within the mean locations on either side of the particular displacement behaves like a rigid body while the remainder of the element does not participate in the motion.

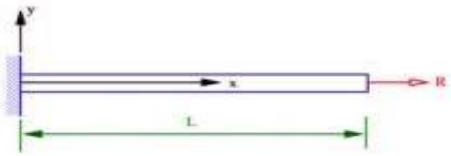
Thus, this assumption excludes the dynamic coupling that exists between the element displacements and hence the resulting element mass matrix is purely diagonal and is called the **"lumped"** mass matrix.

Natural frequencies and mode shapes

The oscillatory motion occurs at certain frequencies known as natural frequencies or characteristic values, and it follows well defined deformation patterns known as mode shapes and characteristic modes.

AXIALLY LOADED BAR

A. Constant End Load



Given: Length L, Section Area A, Young's modulus E
Find: stresses and deformations.

Assumptions:
The cross-section of the bar does not change after loading.
The material is linear elastic, isotropic, and homogeneous.
The load is centric.
End-effects are not of interest to us.

Strength of Materials Approach

From the **equilibrium equation**, the axial force at a random point x along the bar is:

$$\mathbf{f}(x) = \mathbf{R}(= const) \Rightarrow \sigma(x) = \frac{\mathbf{R}}{A}$$

From the **constitutive equation (Hooke's Law)**:

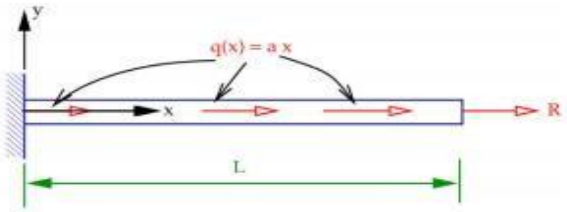
$$\epsilon(x) = \frac{\sigma(x)}{E} = \frac{\mathbf{R}}{AE}$$

Hence, the deformation $\delta(x)$ is obtained from kinematics as:

$$\epsilon = \frac{\delta(x)}{x} \Rightarrow \delta(x) = \frac{\mathbf{R}x}{AE}$$

Note: The stress & strain is independent of x for this case of loading.

B. Linearly Distributed Axial + Constant End Load



From the **equilibrium equation**, the axial force at random point x along the bar is:

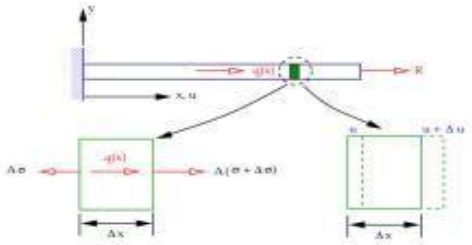
$$\mathbf{f}(x) = \mathbf{R} + \frac{aL + ax}{2}(L - x) = \mathbf{R} + \frac{a(L^2 - x^2)}{2} (\text{ depends on x})$$

In order to now find stresses & deformations (which depend on x) we have to repeat the process for every point in the bar. This is computationally inefficient.

From the **equilibrium equation**, for an infinitesimal element:

$$A\sigma = \mathbf{q}(x)\Delta x + A(\sigma + \Delta\sigma) \Rightarrow A \underbrace{lim}_{\Delta x \to 0} \frac{\Delta\sigma}{\Delta x} + \mathbf{q}(x) = 0 \Rightarrow A\frac{d\sigma}{dx} + \mathbf{q}(x) = 0$$

$$\text{Also, } \epsilon = \frac{d\mathbf{u}}{dx}, \sigma = E\epsilon, \mathbf{q}(x) = ax \Rightarrow AE\frac{d^2\mathbf{u}}{dx^2} + ax = 0$$



Strong Form

$$AE\frac{d^2\mathbf{u}}{dx^2} + ax = 0$$

$\mathbf{u}(0) = 0$ essential BC

$\mathbf{f}(L) = \mathbf{R} \Rightarrow AE\left.\frac{d\mathbf{u}}{dx}\right|_{x=L} = \mathbf{R}$ natural BC

Analytical Solution

$$\mathbf{u}(x) = \mathbf{u}_{hom} + \mathbf{u}_p \Rightarrow \mathbf{u}(x) = C_1 x + C_2 - \frac{ax^3}{6AE}$$

C_1, C_2 are determined from the BC

An analytical solution cannot always be found

Approximate Solution - The Galerkin Approach (#3): Multiply by the weight function $\mathbf{w}$ and integrate over the domain

$$\int_0^L AE\frac{d^2\mathbf{u}}{dx^2}\mathbf{w}dx + \int_0^L ax\mathbf{w}dx = 0$$

Apply integration by parts

$$\int_0^L AE\frac{d^2\mathbf{u}}{dx^2}\mathbf{w}dx = \left[AE\frac{d\mathbf{u}}{dx}\mathbf{w}\right]_0^l - \int_0^L AE\frac{d\mathbf{u}}{dx}\frac{d\mathbf{w}}{dx}dx \Rightarrow$$

$$\int_0^L AE\frac{d^2\mathbf{u}}{dx^2}\mathbf{w}dx = \left[AE\frac{d\mathbf{u}}{dx}(L)\mathbf{w}(L) - AE\frac{d\mathbf{u}}{dx}(0)\mathbf{w}(0)\right] - \int_0^L AE\frac{d\mathbf{u}}{dx}\frac{d\mathbf{w}}{dx}dx$$

But from BC we have $u(0) = 0$, $AE\frac{du}{dx}(L)\mathbf{w}(L) = \mathbf{R}\mathbf{w}(L)$, therefore the approximate weak form can be written as

$$\int_0^L AE\frac{d\mathbf{u}}{dx}\frac{d\mathbf{w}}{dx}dx = \mathbf{R}\mathbf{w}(L) + \int_0^L ax\mathbf{w}dx$$

In Galerkin's method we assume that the approximate solution, $\mathbf{u}$ can be expressed as

$$\mathbf{u}(x) = \sum_{j=1}^{n} u_j N_j(x)$$

$\mathbf{w}$ is chosen to be of the same form as the approximate solution (but with arbitrary coefficients w_i),

$$\mathbf{w}(x) = \sum_{i=1}^{n} w_i N_i(x)$$

Plug $\mathbf{u}(x)$,$\mathbf{w}(x)$ into the approximate weak form:

$$\int_0^L AE \sum_{j=1}^{n} u_j \frac{dN_j(x)}{dx} \sum_{i=1}^{n} w_i \frac{dN_i(x)}{dx} dx = \mathbf{R} \sum_{i=1}^{n} w_i N_i(L) + \int_0^L ax \sum_{i=1}^{n} w_i N_i(x) dx$$

w_i is arbitrary, so the above has to hold $\forall\ w_i$:

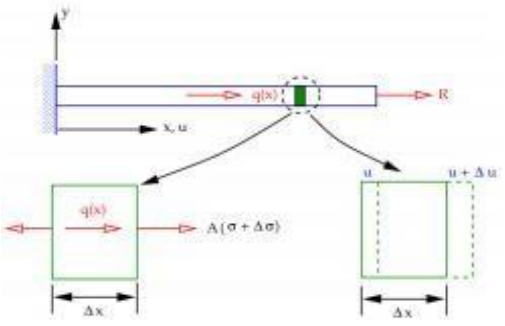
$$\sum_{j=1}^{n} \left[\int_0^L \frac{dN_j(x)}{dx} AE \frac{dN_i(x)}{dx} dx \right] u_j = \mathbf{R} N_i(L) + \int_0^L ax N_i(x) dx \quad i = 1 \ldots n$$

which is a system of n equations that can be solved for the unknown coefficients u_j.

The matrix form of the previous system can be expressed as

$$\mathbf{K}_{ij} u_j = f_i \text{ where } K_{ij} = \int_0^L \frac{dN_j(x)}{dx} AE \frac{dN_i(x)}{dx} dx$$
$$\text{and } f_i = \mathbf{R} N_i(L) + \int_0^L ax N_i(x) dx$$

Finite Element Solution - using 2 discrete elements, of length h (3 nodes)
From the iso-parametric formulation we know the element stiffness matrix $\mathbf{K}^e = \frac{AE}{h} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$. Assembling the element stiffness matrices we get:



$$\mathbf{K}^{tot} = \begin{bmatrix} K_{11}^e & K_{12}^1 & 0 \\ K_{12}^1 & K_{22}^1 + K_{11}^2 & K_{12}^2 \\ 0 & K_{12}^2 & K_{22}^2 \end{bmatrix} =$$

$$\mathbf{K}^{tot} = \frac{AE}{h} \begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}$$

We also have that the element load vector is

$$f_i = \mathbf{R}N_i(L) + \int_0^L axN_i(x)dx$$

Expressing the integral in iso-parametric coordinates $N_i(\xi)$ we have:

$$\frac{d\xi}{dx} = \frac{2}{h}, x = N_1(\xi)x_1^e + N_2(\xi)x_2^e, \Rightarrow$$

$$f_i = \mathbf{R}|_{i=4} + \int_0^L a(N_1(\xi)x_1^e + N_2(\xi)x_2^e)N_i(\xi)\frac{2}{h}d\xi$$

REFERENCE BOOKS

1. BhavikattiS.S., “Finite Element Analysis”, New Age Publishers, 2007.
2. KrishnamoorthyC.S., “Finite Element Analysis: Theory and Programming”, Tata McGraw-Hill, 1995.
3. David Hutton, “Fundamentals of Finite Element Analysis”, Tata McGraw Hill PublishingCompany Limited, New Delhi, 2005.
4. Bathe, K.J., “Finite Element Procedures in Engineering Analysis”, Prentice Hall Inc., 1996.
5. Zienkiewicz, O.C. and Taylor, R.L., “The Finite Element Method”, McGraw Hill, 1987.
6. Chandrupatla, R.T. and Belegundu, A.D., “Introduction to Finite Elements in Engineering”, Prentice Hall of India, 1997.
7. Moaveni, S., “Finite Element Analysis Theory and Application with ANSYS”, Prentice Hall Inc.,1999.
8. http://www.mech.port.ac.uk/sdalby/mbm/CTFRProg.htm.9.http://www.me.mtu.edu/~bettig/MEEM4405