F-DEPFI
N.5
1979
EJ.3

DE ESTUDIOS DE POSGRADO
FACULTAD DE INGENIERIA U.N.A.M.

FINITE ELEMENT METHOD THEORY AND APPLICATION

S. VALLIAPPAN

Sección de Mecánica de Suelos
Noviembre de 1979

5

D-18

FINITE ELEMENT METHOD
THEORY AND APPLICATION

S. VALLIAPPAN *

* Professor. Department of Civil Engineering, The University of New South Wales, Australia

BIBLIOTECA DE LA UNIVERSIDAD NACIONAL
DE
MEXICO
DEPFI

F
DEPFI
U.5
1979
E13

Dentro del Programa de Superación Académica de la Sección de Mecánica de Suelos de la DEPFI, se invitó al doctor S. Valliappan, Profesor del Departamento de Ingeniería Civil de la Universidad de New South Wales de Australia, a impartir un seminario sobre "El método de elementos finitos, teoría y aplicación", mismo que se llevó a cabo en el primer semestre de 1977. El profesor Valliappan es un especialista destacado en el campo del análisis numérico con más de 33 artículos y tres libros escritos en colaboración con renombrados profesores sobre el mismo tema. Los apuntes que ahora se publican fueron elaborados por el doctor Valliappan durante su estancia en la DEPFI y constituyeron el material del seminario.

J. Abraham Díaz Rodríguez,
Jefe de la Sección de Mecánica de Suelos

Noviembre de 1979

G(2) 2456

2456

TABLE OF CONTENTS

CHAPTER 1. INTRODUCTION

1.1 HISTORICAL BACKGROUND 1
1.2 APPLICATIONS OF FEM 2
1.3 METHODS OF ANALYSIS 3
1.4 DESCRIPTION OF FEM 4
1.5 BASIC STEPS IN FE ANALYSIS 5

CHAPTER 2. MATHEMATICAL FORMULATION

2.1 DIRECT FORMULATION - DISPLACEMENT METHOD 8
2.2 CRITERIA FOR CONVERGENCE 11

CHAPTER 3. DISCRETIZATION OF A CONTINUUM

3.1 FINITE ELEMENT DIVISION 21
3.2 REPRESENTATION OF INFINITE MEDIA 22
3.3 BOUNDARY CONDITIONS 22
3.4 EXTERNAL LOADS 23
3.5 CALCULATION OF STRESSES AND STRAINS 24

CHAPTER 4. FORMULATION OF FINITE ELEMENT EQUATIONS - SINGLE ELEMENT

4.1 INTRODUCTION 25
4.2 TRIANGULAR ELEMENT - CONSTANT STRAIN - PLANE STRESS 26
4.3 STRAIN-DISPLACEMENT RELATIONSHIP 35
4.4 STRESS-STRAIN RELATIONSHIP-LINEAR ELASTICITY 37

4.4.1 SPECIAL CASES 39
4.4.2 TRANSFORMATION OF MATERIAL CONSTANTS FROM LOCAL TO GLOBAL SYSTEM 42

TABLE OF CONTENTS

CHAPTER 5. ELEMENT SHAPE FUNCTIONS - ISOPARAMETRIC ELEMENTS

5.1 INTRODUCTION 50
5.2 LAGRANGIAN INTERPOLATION 51
5.3 HERMITIAN INTERPOLATION 53
5.4 NATURAL (LOCAL) COORDINATE SYSTEM 54
5.4.1 TRIANGULAR CO-ORDINATES OR AREA CO-ORDINATES 57
5.5 INTERPOLATION FUNCTIONS 60
5.6 ISOPARAMETRIC ELEMENTS 67
5.6.1 CONVERGENCE CRITERIA 69

CHAPTER 6. ELEMENT EQUATIONS - LINEAR ELASTICITY

6.1 INTRODUCTION 71
6.2 TRANSFORMATION IN LOCAL CO-ORDINATES 72
6.3 NUMERICAL INTEGRATION 75
6.4 EQUIVALENT NODAL FORCES 79
6.5 AXISYMMETRIC PROBLEMS WITH NONAXISYMMETRIC LOADS 81

CHAPTER 7. NONLINEAR ANALYSIS - SOLUTION TECHNIQUES

7.1 INTRODUCTION 87
7.2 VARIABLE STIFFNESS METHOD 88
7.3 INITIAL STIFFNESS METHOD 89
7.3.1 INITIAL STRAIN APPROACH 91
7.3.2 INITIAL STRESS APPROACH 93

TABLE OF CONTENTS

CHAPTER 8. NONLINEAR ANALYSIS - MATERIAL NONLINEARITY

8.1 INTRODUCTION 99
8.2 PLASTICITY 99

8.2.1 YIELD CRITERION 99
8.2.2 PLASTIC POTENTIAL 101
8.2.3 SPECIAL FORMS OF YIELD CRITERIA 102
8.2.4 ELASTIC-PLASTIC STRESS-STRAIN RELATIONSHIP 104

8.3 TENSION CUT-OFF 109
8.4 CREEP 110

8.4.1 CREEP LAWS 111
8.4.2 SELECTION OF TIME INCREMENT 112

8.5 NONLINEAR ELASTICITY 112

FINITE ELEMENT METHOD
THEORY AND APPLICATION

1. INTRODUCTION

1.1 HISTORICAL BACKGROUND

The finite element method (FEM) has become a powerful numerical technique for solving complex problems in science and engineering, mainly due to the advances made earlier in the numerical methods particularly in matrix methods as well as due to the rapid introduction of high speed computers in the market. However, the introduction of concepts and applications of FEM dates back to the era of mathematicians who tried to calculate the perimeter and area of a circle by idealizing it as a regular polygon. It is also interesting to note that the bound solutions which are often discussed in FEM can be traced back to the solution of the area of a circle. If the circle is modelled with an inscribed polygon, a lower bound solution is obtained whereas an upper bound solution is obtained by replacing the circle by a circumscribed polygon. Even though the basic concepts of FEM existed for over two thousand years, for all practical purposes, one can only say that these concepts were actually used for solving physical problems in 1950s by the aeronautical engineers.

In 1956, Turner et al (Ref 1) presented the stiffness analysis for the complex structures, which is the starting point in the rediscovery of FEM. Nevertheless, Clough (Ref 2) was the one who actually used the term FEM in 1960. Since then, a tremendous amount of research has been done in this field and

quite a large number of papers have been published in almost all the journals related to all fields of engineering as well as some in the fields of mathematics and science. In addition, several conferences have been held all over the world and hundreds of papers have been presented in each. The theory and application of FEM have also been presented in numerous text books (Ref 3-22). In order to help the research workers in tracing the references required for their particular work several bibliographics have either been published or under preparation, among them notably Ref (23) is a good source of information.

1.2 APPLICATIONS OF FEM

The FEM is applicable to a variety of boundary value and initial value problems in engineering as well as applied science. Some of these applications are:

1. Stress analysis of structures, stability of structures, dynamic response of structures, thermal stress analysis, torsion of prismatic members
2. Stress analysis of geomechanics problems, soil-structure interaction, slope stability problems, soil dynamics and earthquake engineering, seepage in soils and rocks, consolidation settlement
3. Solutions in fluid mechanics, harbour oscillations, pollution studies, sedimentation
4. Analysis of nuclear reactor structures
5. Stress analysis and flow problems in biomechanics
6. Characteristic study of composites in fibre technology
7. Wave propagation in geophysics
8. Field problems in electrical engineering

Apart from the above mentioned areas, the FEM is also applicable to any other problem as long as the analyst makes certain that the problem is amenable to solution based on the assumptions introduced in the formulation of FEM and appropriate material properties can be provided in a realistic manner.

1.3 METHODS OF ANALYSIS

In general, there are four basic methods of analysis in FEM- displacement method, equilibrium method, mixed method and hybrid method. The field variables or unknown quantities in each of these methods are as follows.

Displacement method - displacements and their derivatives
Equilibrium method - stress components
Mixed method - some displacements and some stress components
Hybrid method - displacements or boundary forces

In the displacement method, smooth displacement distribution is assumed within an element, interelement compatibility of displacement is generally assured and minimum potential energy criterion is used in the formulation.

In the equilibrium method, the interior stress distribution is assumed to be smooth, the equilibrium of boundary tractions is maintained and the minimum complimentary energy is the basis for the formulation.

In the mixed method which is generally used for plate and shell problems, both displacements and stresses are assumed smooth

in the interior, the displacement components and the equivalent stress components are considered to be continuous at the inter-element boundaries and the formulation is based on Reissner's principle.

In the hybrid method, depending on whether the model is displacement type or equilibrium type, the distribution of displacements or stresses within the element is considered to be smooth and along the interelement boundary either assumed compatible displacements or assumed equilibrating boundary tractions are ensured and either modified complementary energy or modified potential energy principle is adopted for the formulation.

Among these four methods, the displacement method is the most widely used approach. However, for plate bending problems either the equilibrium or mixed method is preferred and for some field problems hybrid method is more suitable.

1.4 DESCRIPTION OF FEM

A structure, continuum or a domain is divided into a number of arbitrary shaped parts or regions known as *elements*. These elements are interconnected at joints known as *nodes*. The principal unknown is termed as the *field variable*. This field variable can be displacement, temperature, pore-pressure or stress. The distribution of the field variable within an element is approximated by the use of certain polynomial functions. Variational methods or residual methods are employed

to develop the finite element equations which relate the field variables at the nodes to the corresponding action vector at the nodes of the element. This relationship is provided by the so called property matrix which is based on the material and the geometric properties of the element. Finally these finite element equations are assembled to form a system of algebraic equations for the entire domain. The unknown field variable is obtained by solving this system of algebraic equations.

1.5 BASIC STEPS IN FE ANALYSIS

The basic steps in the finite element analysis of general problems are as follows.

1. The continuum is divided into finite elements of any arbitrary shape.
2. A suitable polynomial is chosen to represent the distribution of the field variable within an element in terms of its nodal values. Thus, the field variables at the nodes become the primary unknowns.
3. Using variational methods or residual methods, the finite element equations are formulated.
4. The individual finite element equations obtained in step 3 are assembled to form a set of algebraic equations for the overall continuum.
5. The solution of the algebraic equations obtained in step 4 yields the values of the field variables at the nodes.
6. From the field variables at the nodes, the secondary variables such as stress, strain for an element can be obtained.

REFERENCES

1. TURNER, M. J., CLOUGH, R. W., MARTIN, H. C., and TOPP, L. J., "Stiffness and deflection analysis of complex structures", J. Aero, Sci., Vol. 23, No. 9, 1956, pp 805-823

2. CLOUGH, R. W., "The finite element method in plane stress analysis", Proc. 2nd ASCE Conf. on Electronic Computation, Pittsburgh, 1960, pp 345-378

3. ZIENKIEWICZ, O. C. and CHEUNG, Y. K., The Finite Element Method in Structural and Continuum Mechanics, McGraw-Hill, London, 1967

4. ZIENKIEWICZ, O. C., The Finite Element Method in Engineering Science, McGraw-Hill, London, 1971

5. SMITH, G. N., An Introduction to Matrix and Finite Element Methods in Civil Engineering, Applied Science, London, 1971

6. DESAI, C. S. and ABEL, J. F., Introduction to the Finite Element Method, Van Nostrand and Reinhold, New York, 1972

7. ODEN, J. T., Finite Elements of Nonlinear Continua, McGraw-Hill, New York,1972

8. URAL, O., Finite Element Method, Intext Educational Publishers, New York, 1973

9. MARTIN, H. C. and CAREY, G. F., Introduction to Finite Element Analysis, McGraw-Hill, New York, 1973

10. STRANG, G. and FIX, G. J., An Analysis of the Finite Element Method, Prentice Hall, N. J., 1973

11. BREBBIA, C. A. and CONNOR, J. J., Fundamentals of Finite Element Technique, Butterworths, London, 1973

12. NORRIS, D. H. and de VRIES, G., The Finite Element Method-Fundamentals and Applications, Academic Press, New York, 1973

13. COOK, R. D., Concepts and Applications of Finite Element Analysis, John Wiley, New York, 1974

14. WACHPRESS, E. L., A Rational Finite Element Basis, Academic Press, New York, 1975

15. FENNER, R. T., Finite Element Method for Engineers, MacMillan Press, London, 1975

16. GALLAGHER, R. H., Finite Element Analysis-Fundamentals, Prentice-Hall, N. J., 1975

17. HUEBNER, K. H., The Finite Element Method For Engineers, John Wiley, New York, 1975

18. ROCKEY, K. C., et al, The Finite Element Method, Crosby, Lockwood, Staples, London, 1975

19. CONNOR, J. J. and BREBBIA, C. A., Finite Element Techniques for Fluid Flow, Butterworths, London, 1976

20. ODEN, J. J. and REDDY, J. N., An Introduction to Mathematical Theory of Finite Elements, John Wiley, New York, 1976

21. SEGERLIND, L. J., Applied Finite Element Analysis, John Wiley, New York, 1976

22. BATHE, K. J. and WILSON, E. L., Numerical Methods in Finite Element Analysis, Prentice-Hall, N. J., 1976

23. NORRIE, D. H. and de VRIES, G., "A Finite Element Bibliography (3 Parts), Report No. 57, Mechanical Engineering Department, The University of Calgary, Canada, 1974

2. MATHEMATICAL FORMULATION

2.1 DIRECT FORMULATION - DISPLACEMENT METHOD

The simplest formulation of finite element equations is probably by directly applying virtual work principle even though the variational principles and residual methods may be more elegant. As pointed out previously, the displacement method is widely used in FE Analysis and hence in this section the mathematical formulation will be based on the displacement model.

The first step in the analysis is to divide the continuum into finite elements. For the present discussion, let us assume some arbitrary shape. The field variable for the displacement model is *displacements*. Hence let us define the displacements at any point within an element in terms of the nodal values as

$$\{u\} = [N]^T \{\delta\}^e \tag{2.1}$$

where $\{u\}$ is the displacement vector at any point within an element

$\{\delta\}^e$ is the nodal displacement vector for the element

and $[N]$ is the shape function defined by a suitable polynomial

The strains can be determined from the nodal displacements as

$$\{\varepsilon\} = [B] \{\delta\}^e \tag{2.2}$$

where $[B]$ is the strain-displacement matrix (based on geometric property)

$[B]$ matrix can be written for finite or infinitesimal strain

From the strains, the stresses can be determined as

$$\{\sigma\} = [D]\ \{\varepsilon\} \tag{2.3}$$

where $[D]$ is the matrix representing the constitutive relationship for the material

$[D]$ matrix can be written either for isotropic or anisotropic material and either for linear or nonlinear material behaviour

If it is desired to include the initial strains which may be due to temperature effects etc and the initial stresses which may be the prestress existing in geomechanical problems, then Eq. (2.3) can be written in a general form as

$$\{\sigma\} = \{\sigma_o\} + [D]\ (\{\varepsilon\} - \{\varepsilon_o\}) \tag{2.4}$$

Let us also define the nodal forces which are statically equivalent to the boundary stresses and distributed loads on the element as $\{F\}^e$. The distributed loads are defined as these acting on an unit volume of material within an element and denoted by $\{p\}$.

In order to make the nodal forces statically equivalent to the actual boundary stresses and distributed loads, let us apply a virtual displacement, $d\{\delta\}^e$ at the nodes.

Then $$d\{u\} = [N]^T\ d\{\delta\}^e \tag{2.5}$$

$$d\{\varepsilon\} = [B]\ d\{\delta\}^e \tag{2.6}$$

External work done = $$W = \left[d\{\delta\}^e\right]^T \{F\}^e \tag{2.7}$$

Internal work done per unit volume is

$$U = d\{\varepsilon\}^T \{\sigma\} - d\{u\}^T \{p\}$$

$$= \left[d\{\delta\}^e\right]^T \left([B]^T \{\sigma\} - [N]^T \{p\}\right) \qquad (2.8)$$

Equating the external work with the internal work

$$\left[d\{\delta\}^e\right]^T \{F\}^e = \left[d\{\delta\}^e\right]^T \left(\int_V [B]^T \{\sigma\}\, dV - \int_V [N]^T \{p\}\, dV\right) \qquad (2.9)$$

Eq. (2.9) is valid for any value of virtual displacement and hence can be written as

$$\{F\}^e = \int_V [B]^T \{\sigma\}\, dV - \int_V [N]^T \{p\}\, dV \qquad (2.10)$$

Or, substituting for $\{\sigma\}$ and $\{\varepsilon\}$ from (2.4) and (2.2)

$$\{F\}^e = \int_V \left([B]^T [D] [B]\, dV\right) \{\delta\}^e - \int_V [B]^T [D] \{\varepsilon_o\}\, dV +$$

$$+ \int_V [B]^T \{\sigma_o\}\, dV - \int_V [N]^T \{p\}\, dV \qquad (2.11)$$

Using the stiffness approach, Eq. (2.11) can be written as

$$\{F\}^e = [K]^e \{\delta\}^e + \{F\}_i^e + \{F\}_p^e \qquad (2.12)$$

where $\{F\}^e$ = total force vector at the nodes of an element

$[K]^e$ = stiffness matrix for an element

$$= \int_V [B]^T [D] [B]\, dV$$

$\{\delta\}^e$ = displacement vector at the nodes of an element

$\{F\}_i^e$ = initial force vector due to initial strains and initial stresses

$$= - \int_V [B]^T [D] \{\varepsilon_o\} \, dV + \int_V [B]^T \{\sigma_o\} \, dV$$

$\{F\}_p^e$ = nodal force vector due to distributed loads

$$= - \int_V [N]^T \{p\} \, dV$$

After having obtained the stiffness matrix via Eq. (2.12), the remaining steps are as given in Section 1.5.

2.2 CRITERIA FOR CONVERGENCE

From the foregoing discussion on mathematical formulation, it can be seen that three primary relationships (2.1, 2.2 and 2.3) are included in the finite element equations. Among the three relationships, Eq. 2.2 and 2.3 are governed by the principles of continuum mechanics whereas in Eq. 2.1, the selection of the shape function (or displacement function) N is left to the choice of the analyst. However, some restrictions have to be imposed on its choice for the sake of obtaining an exact solution to the problem.

Any numerical technique is approximate to a certain degree and to obtain acceptable solution to a problem, the method should converge to the exact solution. In finite element method, for instance, if the displacement model is chosen it can be shown that in general the element stiffness calculated will be higher

than the true stiffness. Hence the displacements obtained for certain loading will be less than the actual ones and thus a lower bound solution is achieved. For the displacement model, in order to obtain convergence the following requirements have to be satisfied in choosing an appropriate displacement function.

1. The displacement function should be chosen in such a way that it ensures continuity within an element and compatibility at the interface between elements. For example, there should not be gaps, overlaps or discontinuities between two adjacent elements when they deform.

2. The displacement function chosen should be such that it includes the term for rigid body displacement. This should be necessary because no straining will be caused due to rigid body movement.

3. The displacement function should be capable of yielding constant strain condition. This is true because if the finite element division is made smaller and smaller, then constant strain will prevail in an infinitesimal element.

EXAMPLE 2.1

Determine the stiffness matrix for the line element shown in Fig 2.1

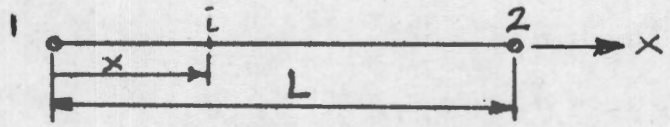


Fig 2.1

a) Using Mechanics of Solids :

Let the displacement of the bar due to an axial load, P, be u

Then, Strain = $\varepsilon = \frac{u}{L}$

Stress = $\sigma = E.\varepsilon. = E.\frac{u}{L}$

Force = $P = \sigma.A. = EA\frac{u}{L}$

Force-Displacement Relationship: $P = K.u.$

where $K = \frac{EA}{L}$

(or) Displacement = $u = \frac{PL}{EA} = f.P.$

b) Using FEM :

Since there are only two nodes, the displacement variation is linear. Thus, the displacement at any point i can be defined using a linear polynomial as

$$u_x = \alpha_1 + \alpha_2 x \qquad (1)$$

The two constants α_1 and α_2 can be determined by substituting the values of u_1 and u_2 at nodes 1 and 2 and their co-ordinates in Eq. (1)

Thus; $u_1 = \alpha_1$

$$u_2 = \alpha_1 + \alpha_2 L$$

$$= u_1 + \alpha_2 L$$

or $$\alpha_2 = \frac{u_2 - u_1}{L}$$

Therefore, $$u = u_1 + \frac{u_2 - u_1}{L} x = u_1 \left[1 - \frac{x}{L}\right] + u_2 \left[\frac{x}{L}\right]$$

or $$u = [N] \{\delta\}^e$$

where $$[N] = \left[1 - \frac{x}{L} \quad \frac{x}{L}\right]$$

$$\{\delta\}^e = \begin{bmatrix} u_1 \\ u_2 \end{bmatrix}$$

$$\text{Strain} = \varepsilon = \frac{\partial u}{\partial x} = [B] \{\delta\}^e = \left[\frac{\partial N}{\partial x}\right] \{\delta\}^e$$

$$= \begin{bmatrix} -1/L & 1/L \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix}$$

$$\text{Stress} = \sigma = E.\varepsilon. = [D] [B] \{\delta\}^e$$

$$= \begin{bmatrix} -E/L & E/L \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix}$$

$$\text{Stiffness Matrix} = [k^e] = \int_V [B]^T [D] [B] \, dV$$

$$= \int_0^L \begin{bmatrix} -1/L \\ 1/L \end{bmatrix} E \begin{bmatrix} -1/L & 1/L \end{bmatrix} A \, dx$$

$$= \frac{EA}{L^2} \int_0^L \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} dx$$

$$= \frac{EA}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

Force-Displacement Relationship:

$$\begin{bmatrix} F_1 \\ F_2 \end{bmatrix} = \frac{EA}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix}$$

If $u_1 = 0$ and $F_2 = P$,

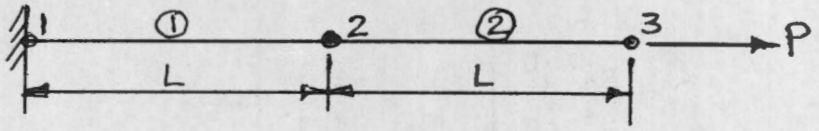
$$F_1 = -\frac{EA}{L} u_2 \quad \text{and} \quad P = \frac{EA}{L} u_2$$

$$\text{or} \quad u_2 = \frac{PL}{EA}$$

$$F_1 = -\frac{EA}{L} \frac{PL}{EA} = -P$$

EXAMPLE 2.2

Determine the displacements at the nodes, the strains and the stresses for the elements of the member shown in Fig 2.2. Both elements have same values of E and A.



From example 2.1,

$$[k_1^e] = [k_2^e] = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$[K] = [k_1^e] + [k_2^e]$$

$$= \left[\begin{array}{c|c|c} 1 & -1 & 0 \\ \hline -1 & 1+1 & -1 \\ \hline 0 & -1 & 1 \end{array}\right]$$

Thus,

$$\begin{bmatrix} F_1 \\ F_2 \\ F_3 \end{bmatrix} = \frac{EA}{L}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix}$$

Boundary Conditions:

$$u_1 = 0 \qquad F_3 = P$$

$$\begin{bmatrix} F_1 \\ 0 \\ P \end{bmatrix} = \frac{EA}{L} \begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix} \begin{bmatrix} 0 \\ u_2 \\ u_3 \end{bmatrix} \qquad \text{I}$$

Solving Eq. I,

$$P = \frac{EA}{L} \left[-u_2 + u_3 \right]$$

$$0 = 2u_2 - u_3 \quad \text{(or)} \quad u_3 = 2u_2$$

$$P = \frac{EA}{L} \left[-u_2 + 2u_2 \right] = \frac{EA}{L} u_2$$

$$u_2 = \frac{PL}{EA}$$

$$u_3 = 2 \frac{PL}{EA}$$

$$F_1 = - \frac{EA}{L} u_2 = - \frac{EA}{L} \frac{PL}{AE} = -P$$

<u>Element 1</u>: Strain = $\varepsilon = [B] \{\delta\}^e = [B] \begin{bmatrix} u_1 \\ u_2 \end{bmatrix}$

$$= \begin{bmatrix} - \frac{1}{L} & \frac{1}{L} \end{bmatrix} \begin{bmatrix} 0 \\ \frac{PL}{EA} \end{bmatrix}$$

$$= \frac{P}{EA}$$

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

$$= E \frac{P}{EA} = \frac{P}{A}$$

Element 2: Strain = $\varepsilon = [B] \begin{bmatrix} u_2 \\ u_3 \end{bmatrix}$

$$= \begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix} \begin{bmatrix} \frac{PL}{EA} \\ 2\frac{PL}{EA} \end{bmatrix}$$

$$= \frac{P}{EA}$$

$$\text{Stress} = \sigma = E\frac{P}{EA} = \frac{P}{A}$$

EXAMPLE 2.3

In the case of the axial member illustrated in Example 2.1, prove the conditions of compatibility, constant strain and rigid body movement.

Suppose the polynomial assumed for defining the variation of displacement is given by

$$u = \alpha_1 x^2 + \alpha_2 x^3$$

will the convergence criteria be satisfied?

Condition 1: Interelement Compatibility.

Element 1: $u_x = \begin{bmatrix} 1 - \frac{x}{L} & \frac{x}{L} \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix}$

At node 2, $x = L$, $u = \begin{bmatrix} 0 & 1 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = u_2$

Element 2:

At node 2 $x = 0$, $u = \begin{bmatrix} 1 & 0 \end{bmatrix} \begin{bmatrix} u_2 \\ u_3 \end{bmatrix} = u_2$

Thus, compatibility is satisfied.

Condition 2: Constant Strain Criterion.

$$\varepsilon = \frac{\partial u}{\partial x} = \alpha_2$$

Since α_2 is a constant, this condition is satisfied.

Condition 3: Rigid Body Movement.

In the polynomial, $u = \alpha_1 + \alpha_2 x$ the rigid body movement can be prescribed by putting $x = 0$,

Thus $u = \alpha_1$

Then strain, $\varepsilon = \frac{\partial u}{\partial x} = 0$

Since no straining takes place due to rigid body movement, Condition 3 is satisfied.

If $u = \alpha_1 x^2 + \alpha_2 x^3$

$$\varepsilon = \frac{\partial u}{\partial x} = 2\alpha_1 x + 3\alpha_2 x^2$$

Since the strain depends on the co-ordinates of the point under consideration, the condition of constant strain is violated.

For rigid body movement, there is no strain. Thus, if we equate the strain to zero

$$2\alpha_1 x + 3\alpha_2 x^2 = 0$$

$$\alpha_2 = -\frac{2\alpha_1}{3x}$$

$$u = \alpha_1 x^2 - \frac{2\alpha_1}{3} x^2 = \frac{1}{3} \alpha_1 x^2$$

The displacement is not constant. It is a function of co-ordinate and hence Condition 3 is violated.

3. DISCRETIZATION OF A CONTINUUM

3.1 FINITE ELEMENT DIVISION

The more commonly asked questions regarding the division of a structure or a continuum into finite elements are 1) how many elements are necessary to obtain a practically acceptable solution to the problem? 2) should the division be a regular pattern or irregular? and 3) how does one check about the convergence of the solution? The real answer to all these questions is exercising intuition, experience and engineering judgement. Nevertheless, certain guidelines can be put forth for the sake of novices.

Regarding the total number of elements required for a problem, one must try to use as many elements as required for the desired accuracy bearing in mind that too many elements will result in enormous computer time. The division of the structure itself depends on the complexity of loading, nonhomogeneous nature of the structure, geometrical discontinuities and irregularities. As far as possible, it is better to use a graded mesh in such a way that finer divisions are provided near the areas of stress concentration and coarser elements are used at places of flat stress gradients. Most of the problems which require the use of finite element technique for obtaining the solutions are so complex that they cannot be solved by anyother analytical technique to yield an exact solution. Hence, one should use his engineering judgement to assess the accuracy of the results

or solve the problem by using one coarser and one finer mesh division in order to check the convergence.

Apart from the points mentioned above, one must also take care in labelling the nodes and elements such that the bandwidth of the stiffness matrix is kept to a minimum in order to reduce the computer time for solving the simultaneous equations.

3.2 REPRESENTATION OF INFINITE MEDIA

In most of the structural problems, the boundaries of the structures analysed by FEM are clearly defined. Whereas in geomechanical problems, invariably one encounters infinite media to be represented by a finite model. In the case of static problems, it is necessary to use either engineering judgement or the experience of other investigators in fixing the extent of the model to be analysed. Generally, the influence of disturbance or loaded area extends only to a certain distance from the source of disturbance. In the case of dynamic problems, one has to resort to the use of special boundary conditions.

3.3 BOUNDARY CONDITIONS

For any physical problem, appropriate boundary conditions have to be specified. Otherwise, the structure will be free to experience any amount of rigid body motion. Mathematically speaking, if the boundary constraints are not imposed, then the stiffness matrix will be singular, that is, it cannot be inverted.

Basically there are two types of boundary conditions - geometric and natural. If an appropriate formulation is used on the basis of variational principles for the finite element equations, then the natural boundary conditions are automatically satisfied. The geometric boundary conditions can be divided into homogeneous and nonhomogeneous conditions. If the displacements (for the displacement method) are specified as zero, then the boundary condition is known as homogeneous whereas it is nonhomogeneous when the displacements are not zero at the boundary. Further the boundary condition can be classified as normal or skew. If the boundaries are parallel to global axes, the conditions at those boundaries are normal. If the boundaries are not parallel to global axes, then the conditions at those boundaries are called skew boundary conditions. Special techniques have to be employed to treat the skew boundary conditions.

3.4 EXTERNAL LOADS

In FEM, since the element stiffness matrix is formulated to relate the nodal displacements to the corresponding nodal forces, all the external forces should be applied only at the nodes. This involves calculating the statically equivalent nodal forces for the distributed loads and surface tractions on the basis of formulation used for element stiffness matrix. This is in fact known as consistent loads which is superior to lumped loads which is actually alloting the loads to the various nodes on the basis of intuition.

3.5 CALCULATION OF STRESSES AND STRAINS

After evaluating the stiffness matrix for the whole structure, with the use of appropriate boundary conditions and external loads, the displacements at all nodes can be determined. From the nodal displacements of an element, the strains and the stresses at any point within an element can be calculated. It is always advisable to determine the stresses at a point within an element instead of averaging it at a nodal point, from the stress values contributed by the surrounding elements.

4. FORMULATION OF FINITE ELEMENT EQUATIONS - SINGLE ELEMENT

4.1 INTRODUCTION

The various shapes which have been used to discretize the continuum into finite elements are: line elements for one dimension, triangular, rectangular and quadrilateral elements for two dimension and tetrahedron, triangular and rectangular prism elements for three dimension. Again, these elements may have either straight or curved sides and may have only corner nodes or corner nodes and either interior nodes along the boundaries of an element or interior nodes within an element. Among all these elements, the simplest shape which can be visualized is probably the triangle because complex boundaries can be fairly approximated with the sides of the triangle. This is one of the reasons why the triangular element was developed first by Turner et al. The other reason is probably that the structural engineers are quite familiar with the triangular shape since it is often encountered in the analysis of framed structures. Thus, in this chapter, the simplest formulation which was originally devised for the triangular element will be presented. Later, the general formulation for simple as well as complex elements will be presented on the basis of isoparametric concept.

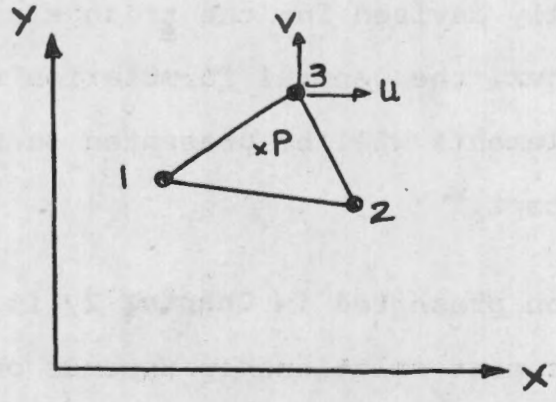
From the discussion presented in Chapter 2, it can be seen that the strain-displacement relationship depends only on the finite strain or infinitesimal strain theory and not on the type of

finite element. Similarly, the stress-strain relationship depends only on the linear elastic theory or nonlinear theory (plasticity, creep, visco-elasticity, visco-plasticity) and not on the element shape. The only criteria which is different for each type of element is the shape function or displacement function.

In the following sections, the complete formulation for the triangular element on the basis of infinitesimal strain, linear elastic theory for plane stress condition will be given first followed by the strain-displacement based on the infinitesimal strain and the stress-strain relationships based on linear elastic theory for two dimensional plane stress/plane strain, axisymmetric and three dimensional cases. The development of shape functions which is the crux of the finite element formulation will be presented in the next chapter.

4.2 TRIANGULAR ELEMENT - CONSTANT STRAIN - PLANE STRESS

Fig 4.1 shows the triangular element (123) in the cartesian co-ordinate system



Let the unknowns for this problem be horizontal displacement, u and vertical displacement, v at each node. Since there are 3 nodes, the number of unknowns is six. On each side, there are only two nodes and hence the possible variation of displacement is linear and therefore the strain within the element is constant.

Displacement Function

The displacements u and v at any point, P, within the element 123 can be uniquely defined by the unknown displacements at the nodes 1, 2 and 3. Since the displacement field is linear, the displacements at point, P, can be represented by linear polynomials.

$$u = \alpha_1 + \alpha_2 x + \alpha_3 y$$
$$v = \alpha_4 + \alpha_5 x + \alpha_6 y \qquad (4.1)$$

The constants in the polynomials, α_1 to α_6 can be evaluated in terms of the nodal co-ordinates and the nodal displacements as follows.

$$u_1 = \alpha_1 + \alpha_2 x_1 + \alpha_3 y_1$$
$$u_2 = \alpha_1 + \alpha_2 x_2 + \alpha_3 y_2 \qquad (4.2)$$
$$u_3 = \alpha_1 + \alpha_2 x_3 + \alpha_3 y_3$$

There are 3 unknowns $(\alpha_1, \alpha_2, \alpha_3)$ and 3 equations given by Eq. (4.2). Hence, solving these equations

$$\alpha_1 = \frac{1}{2\Delta} (a_1 u_1 + a_2 u_2 + a_3 u_3)$$

$$\alpha_2 = \frac{1}{2\Delta} (b_1 u_1 + b_2 u_2 + b_3 u_3) \qquad (4.3)$$

$$\alpha_3 = \frac{1}{2\Delta} (c_1 u_1 + c_2 u_2 + c_3 u_3)$$

where $2\Delta = \det \begin{vmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{vmatrix}$ = 2x area of triangle 123 (4.3a)

$$a_1 = x_2 y_3 - x_3 y_2$$

$$b_1 = y_2 - y_3$$

$$c_1 = x_3 - x_2$$

$$a_2 = x_3 y_1 - x_1 y_3$$

$$b_2 = y_3 - y_1 \qquad (4.3b)$$

$$c_2 = x_1 - x_3$$

$$a_3 = x_1 y_2 - x_2 y_1$$

$$b_3 = y_1 - y_2$$

$$c_3 = x_2 - x_1$$

Now, substituting Eq. (4.3) into Eq. (4.1), and rearranging the terms,

$$u = \frac{1}{2\Delta}\left[(a_1 + b_1x + c_1y)\ u_1 + (a_2 + b_2x + c_2y)\ u_2 + \right.$$

$$\left. + (a_3 + b_3x + c_3y)\ u_3\right] \qquad (4.4)$$

Since the equation for the vertical displacement is similar,

$$v = \frac{1}{2\Delta}\left[(a_1 + b_1x + c_1y)\ v_1 + (a_2 + b_2x + c_2y)\ v_2 + \right.$$

$$\left. + (a_3 + b_3x + c_3y)\ v_3\right] \qquad (4.5)$$

In order to conform with the definition of the field variable in terms of the shape functions, we can rewrite Eqs. (4.4) and (4.5) as follows.

Let the field variable be $\{\emptyset\}$, then

$$\{\emptyset\} = \begin{bmatrix} u \\ v \end{bmatrix} = \begin{bmatrix} N_1 & o & N_2 & o & N_3 & o \\ o & N_1 & o & N_2 & o & N_3 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix} \qquad (4.6)$$

$$= |N|\ \{\delta\}^e$$

where
$$N_1 = \frac{1}{2\Delta}(a_1 + b_1 x + c_1 y)$$
$$N_2 = \frac{1}{2\Delta}(a_2 + b_2 x + c_2 y) \qquad (4.6a)$$
$$N_3 = \frac{1}{2\Delta}(a_3 + b_3 x + c_3 y)$$

Strains

The strains at point, P, within the element 123, can be obtained by differentiating the displacements. Thus, for plane stress conditions

$$\{\varepsilon\} = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \end{bmatrix} \qquad (4.7)$$

Or, using Eq. (4.6)

$$\{\varepsilon\} = \begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial y} & \frac{\partial N_3}{\partial x} \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix} \qquad (4.8)$$

$$\{\varepsilon\} = \frac{1}{2\Delta}\begin{bmatrix} b_1 & o & b_2 & o & b_3 & o \\ o & c_1 & o & c_2 & o & c_3 \\ c_1 & b_1 & c_2 & b_2 & c_3 & b_3 \end{bmatrix}\begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix} \tag{4.8a}$$

$$= [B]\ \{\delta\}^e \tag{4.8b}$$

Stresses

Assuming that the element is of isotropic material, for plane stress conditions, the stress-strain relationship can be written as

$$\varepsilon_x = \frac{\sigma_x}{E} - \nu\frac{\sigma_y}{E}$$

$$\varepsilon_y = \frac{\sigma_y}{E} - \nu\frac{\sigma_x}{E} \tag{4.9}$$

$$\gamma_{xy} = \frac{\tau_{xy}}{G}$$

Or, the stresses can be written in terms of the strains,

$$\{\sigma\} = [D]\ \{\varepsilon\} \tag{4.10}$$

where $[D]$ = Elasticity matrix

$$= \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & o \\ \nu & 1 & o \\ o & o & (1-\nu)/2 \end{bmatrix} \tag{4.10a}$$

in which E is modulus of elasticity and ν is Poisson's ratio.

It is to be noted that for the sake of simplicity, the initial strains and the initial stresses have been omitted from the constitutive relation given in Eq. (4.10).

Stiffness Matrix

The stiffness matrix given by Eq. (2.12)

$$[K]^e = \int_V [B]^T [D] [B] \, dV$$

can be written as

$$[K]^e = [B]^T [D] [B] \, t\Delta \tag{4.11}$$

if the thickness, t, is constant over the entire area and since the matrices $[B]$ and $[D]$ do not depend on the co-ordinates x and y, the integral over the area can be represented by Δ, which is the area of the triangle 123 defined by Eq. (4.3a).

The expanded version of $[K]^e$ can be written as

$$[K]^e = \begin{bmatrix} K_{11} & K_{12} & K_{13} \\ K_{21} & K_{22} & K_{23} \\ K_{31} & K_{32} & K_{33} \end{bmatrix} \tag{4.12}$$

where K_{11}, K_{12}... are 2 x 2 submatrices

Nodal Forces

Surface Forces: Let $\{T\}$ be the traction vector along the side of the element. Then, the equivalent nodal forces are given by

$$\{F\}^e = \int_S [N]^T \{T\} \, dS \tag{4.13}$$

In this particular case of a triangle under plane stress, assuming thickness to be constant

$$\{F\}^e = t \int_o^l [N]^T \{T\}\, dx \tag{4.14}$$

where l is the length of the side under consideration and {T} consists of X and Y forces.

Distributed Body Forces: The distributed body forces may be due to self weight. Or in a general case, the components in x and y directions are X and Y. Denoting these in vector form as {p}, the equivalent nodal forces for the triangular element are

$$\begin{aligned} \{F\}_p^e &= - \int_V [N]^T \{p\}\, dV \\ &= - t \int [N]^T \begin{bmatrix} X \\ Y \end{bmatrix} dx\, dy \end{aligned} \tag{4.15}$$

Assuming the body forces to be constant throughout the element,

$$\{F\}_p^e = - t \begin{bmatrix} X \\ Y \end{bmatrix} \int [N]^T dx\, dy \tag{4.16}$$

Again, for the sake of simplicity, if we assume the origin of the co-ordinates at the centroid of the element

then $\int x\, dx\, dy = \int y\, dx\, dy = 0$

$$\text{Therefore, } \{F_1\}_p = - t \begin{bmatrix} X \\ Y \end{bmatrix} \frac{1}{2\Delta} \int a_1 \, dx \, dy \tag{4.17}$$

$$= - \frac{a_1 \, t}{2} \begin{bmatrix} X \\ Y \end{bmatrix}$$

$$\text{Since } a_1 = \frac{2\Delta}{3} = a_2 = a_3$$

$$\{F_1\}_p = - \frac{t}{2} \frac{2\Delta}{3} \begin{bmatrix} X \\ Y \end{bmatrix} = - \frac{t\Delta}{3} \begin{bmatrix} X \\ Y \end{bmatrix} \tag{4.18}$$

Assuming the thickness to be unity, for the whole element

$$\{F\}_p^e = - \frac{\Delta}{3} \begin{bmatrix} X \\ Y \\ X \\ Y \\ X \\ Y \end{bmatrix} \tag{4.19}$$

Eq. (4.19) implies distributing the body forces equally at the three nodes.

4.3 STRAIN-DISPLACEMENT RELATIONSHIP

The nonlinear theory of elasticity or the theory of finite deformations is quite different from the theory of elasticity due to the fact that the former accounts for the difference between the geometry of deformed and undeformed states whereas the latter neglects it. There are different methods of describing the deformed state such as Lagrangian Method and Eulerian Method. For the purpose of present discussion, Lagrangian Method will be adopted. In this method, the co-ordinates of points under consideration in the undeformed state coincide with co-ordinates of the deformed state. Thus, the strain-displacement relation in the cartesian co-ordinate system can be written as

$$\varepsilon_x = \frac{\partial u}{\partial x} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial x}\right)^2 + \left(\frac{\partial w}{\partial x}\right)^2\right]$$

$$\varepsilon_y = \frac{\partial v}{\partial y} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial y}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial y}\right)^2\right] \qquad (4.20)$$

$$\varepsilon_z = \frac{\partial w}{\partial z} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial z}\right)^2 + \left(\frac{\partial v}{\partial z}\right)^2 + \left(\frac{\partial w}{\partial z}\right)^2\right]$$

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

$$\gamma_{yz} = \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} + \frac{\partial u}{\partial y}\frac{\partial u}{\partial z} + \frac{\partial v}{\partial y}\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\frac{\partial w}{\partial z}$$

$$\gamma_{zx} = \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} + \frac{\partial u}{\partial z}\frac{\partial u}{\partial x} + \frac{\partial v}{\partial z}\frac{\partial v}{\partial x} + \frac{\partial w}{\partial z}\frac{\partial w}{\partial x}$$

In the case of small deformations, the infinitesimal strain theory can be assumed and hence Eq. (4.20) can be used by neglecting the higher order terms. Thus, the simplified relationship will be as follows

$$\begin{aligned}
\varepsilon_x &= \frac{\partial u}{\partial x} \\
\varepsilon_y &= \frac{\partial v}{\partial y} \\
\varepsilon_z &= \frac{\partial w}{\partial z} \\
\gamma_{xy} &= \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\
\gamma_{yz} &= \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \\
\gamma_{zx} &= \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}
\end{aligned} \tag{4.21}$$

Eq. (4.21) is applicable for the case of three-dimensional bodies. In two dimensional situations we have only ε_x, ε_y and γ_{xy}.

In the case of axisymmetric solids, the strain-displacement relations are expressed in terms of cylindrical co-ordinates.

$$\begin{aligned}
\varepsilon_r &= \frac{\partial u_r}{\partial r} \\
\varepsilon_z &= \frac{\partial u_z}{\partial z} \\
\varepsilon_\theta &= \frac{u_r}{r} \\
\gamma_{rz} &= \frac{\partial u_r}{\partial z} + \frac{\partial u_z}{\partial r}
\end{aligned} \tag{4.22}$$

4.4 STRESS-STRAIN RELATIONSHIP.- LINEAR ELASTICITY

The general relationship between stresses and strains for an anisotropic body based on Hooke's law can be written as

$$\{\sigma\} = [D]\{\varepsilon\} \tag{4.23}$$

where $\{\sigma\}$ and $\{\varepsilon\}$ have 6 components each and hence $[D]$ has 36 constants. However, due to symmetry ($D_{12} = D_{21}$, $D_{13} = D_{31}$ etc) the number of independent constants can be reduced to 21. For more practical cases, we can further reduce the number of constants to 9 on the basis of three planes of elastic symmetry. Thus the strains in terms of stresses can be written as

$$\begin{aligned}
\varepsilon_x &= \frac{1}{E_x}\sigma_x - \frac{\nu_{yx}}{E_y}\sigma_y - \frac{\nu_{zx}}{E_z}\sigma_z \\
\varepsilon_y &= -\frac{\nu_{xy}}{E_x}\sigma_x + \frac{1}{E_y}\sigma_y - \frac{\nu_{zy}}{E_z}\sigma_z \\
\varepsilon_z &= -\frac{\nu_{xz}}{E_x}\sigma_x - \frac{\nu_{yz}}{E_y}\sigma_y + \frac{1}{E_z}\sigma_z \\
\gamma_{xy} &= \frac{1}{G_{xy}}\tau_{xy} \\
\gamma_{yz} &= \frac{1}{G_{yz}}\tau_{yz} \\
\gamma_{zx} &= \frac{1}{G_{zx}}\tau_{zx}
\end{aligned} \tag{4.24}$$

Even though there are 12 constants in Eq. (4.24), only 9 of these are independent because

$$E_x\,\nu_{yx} = E_y\,\nu_{xy};\quad E_y\,\nu_{zy} = E_z\,\nu_{yz};\quad E_z\,\nu_{xz} = E_x\,\nu_{zx} \tag{4.25}$$

For the case of isotropic material, the number of material parameters will further be reduced to two - modulus of elasticity, E and Poisson's ratio, ν. Thus, Eq. (4.24) can be written in matrix form as

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{bmatrix} = \begin{bmatrix} 1/E & -\nu/E & -\nu/E & & & \\ -\nu/E & 1/E & -\nu/E & & 0 & \\ -\nu/E & -\nu/E & 1/E & & & \\ & & & 1/G & 0 & 0 \\ & 0 & & 0 & 1/G & 0 \\ & & & 0 & 0 & 1/G \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{bmatrix}$$

$$\text{where } G = \frac{E}{2(1+\nu)} \tag{4.26}$$

Eq. (4.26) can be rewritten in order to express stresses in terms of strains. Thus the matrix $[D]$ for three dimensional isotropic case will be

$$|D| = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & & & \\ \nu & 1-\nu & \nu & & 0 & \\ \nu & \nu & 1-\nu & & & \\ & & & \frac{1-2\nu}{2} & 0 & 0 \\ & 0 & & 0 & \frac{1-2\nu}{2} & 0 \\ & & & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \tag{4.27}$$

4.4.1 SPECIAL CASES

Plane Stress.- The plane stress condition is characterized by the presence of stress only in the plane since the dimension normal to the plane is very small. Therefore, the non-zero components of stress are σ_x, σ_y and τ_{xy}. Thus, the stress-strain relations for the isotropic material is given by

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} \tag{4.28}$$

For the case of anisotropic material, Eq. (4.24) can be reduced to

$$\begin{aligned} \varepsilon_x &= \frac{\sigma_x}{E_x} - \frac{\nu_{yx}}{E_y}\sigma_y \\ \varepsilon_y &= \frac{\sigma_y}{E_y} - \frac{\nu_{xy}}{E_x}\sigma_x \\ \tau_{xy} &= \frac{1}{G_{xy}}\tau_{xy} \end{aligned} \tag{4.29}$$

Letting $\frac{E_x}{E_y} = n$ and $\frac{G_{xy}}{E_y} = m$ and expressing stresses in terms of strains, the matrix [D] can be written as

$$[D] = \frac{E_y}{(1-n\,\nu_{yx}^2)} \begin{bmatrix} n & n\nu_{yx} & 0 \\ n\nu_{yx} & 1 & 0 \\ 0 & 0 & m(1-n\nu_{yx}^2) \end{bmatrix} \tag{4.30}$$

Plane Strain.- In the case of plane strain problems, the longitudinal direction is long compared to the other two directions (representing the plane) and hence the strain in that direction vanishes leaving only three components of strain, ε_x, ε_y and γ_{xy}. However, it is to be noted that the stress, $\sigma_z \neq 0$. Thus, for the isotropic material, Eq. (4.26) reduces to

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} 1/E & -\nu/E & -\nu/E & 0 \\ -\nu/E & 1/E & -\nu/E & 0 \\ -\nu/E & -\nu/E & 1/E & 0 \\ 0 & 0 & 0 & 1/G \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \end{bmatrix} \tag{4.31}$$

$$\text{Since } \varepsilon_z = 0,\ \sigma_z = \nu(\sigma_x + \sigma_y) \tag{4.32}$$

Now, the stresses σ_x, σ_y, τ_{xy} can be expressed in terms of strains ε_x, ε_y and γ_{xy} using Eq. (4.32)

$$[D] = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \tag{4.33}$$

For the case of anisotropic material, the matrix [D] can be obtained from Eq. (4.24) as follows

$$\text{let } 1 = (1+\nu_{xy})(1-\nu_{xy} - 2n\nu_{yx}^2)$$

$$[D]=\frac{E_y}{1}\begin{bmatrix} n(1-n\nu_{yx}^2) & n\nu_{yx}(1+\nu_{xy}) & 0 \\ n\nu_{yx}(1+\nu_{xy}) & (1-\nu_{xy}^2) & 0 \\ 0 & 0 & ml \end{bmatrix} \quad (4.34)$$

Axisymmetric.- Axisymmetric problems are those involving solids of revolution around an axis of symmetry (usually z axis). In this case, the non-zero components of stress and strain are σ_r, σ_z, σ_θ, τ_{rz} and ε_r, ε_z, ε_θ and ε_{rz}. For the isotropic material, the matrix $[D]$ can be written as

$$[D] = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1-\nu & \nu & \nu & 0 \\ \nu & 1-\nu & \nu & 0 \\ \nu & \nu & 1-\nu & 0 \\ 0 & 0 & 0 & \frac{(1-2\nu)}{2} \end{bmatrix} \quad (4.35)$$

For the case of anisotropic material, following Eq. (4.24) and (4.34)

$$[D]=\frac{E_y}{1}\begin{bmatrix} n(1-n\nu_{yx}^2) & n\nu_{yx}(1+\nu_{xy}) & n(\nu_{xy}+ n\nu_{yx}^2) & 0 \\ & (1-\nu_{xy}^2) & n\nu_{yx}(1+\nu_{xy}) & 0 \\ & & n(1 - n\nu_{yx}^2) & 0 \\ \text{Symmetric} & & & ml \end{bmatrix} \quad (4.36)$$

4.4.2 TRANSFORMATION OF MATERIAL CONSTANTS FROM LOCAL TO GLOBAL SYSTEM

If the direction of strata is inclined to the global x axis as shown in Fig 4.2, then the matrix [D] can be obtained by transforming the matrix [D'] which is expressed in the local system as follows:

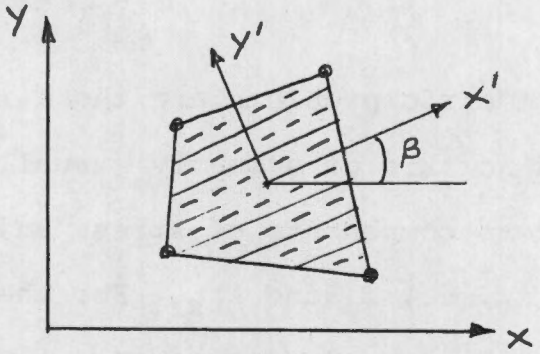


Fig 4.2

Since the work done in both systems should be equal,

$$\{\sigma'\}^T \{\varepsilon'\} = \{\sigma\} \{\varepsilon\} \tag{4.37}$$

$$\text{or, } \{\varepsilon'\}^T [D'] \{\varepsilon'\} = \{\varepsilon\}^T [D] \{\varepsilon\} \tag{4.38}$$

The strains in the inclined system can be expressed in terms of the global system as

$$\{\varepsilon'\} = [T]^T \{\varepsilon\} \tag{4.39}$$

$$\text{where } [T] = \begin{bmatrix} \cos^2\beta & \sin^2\beta & -2\sin\beta\cos\beta \\ \sin^2\beta & \cos^2\beta & 2\sin\beta\cos\beta \\ \sin\beta\cos\beta & -\sin\beta\cos\beta & \cos^2\beta-\sin^2\beta \end{bmatrix} \tag{4.40}$$

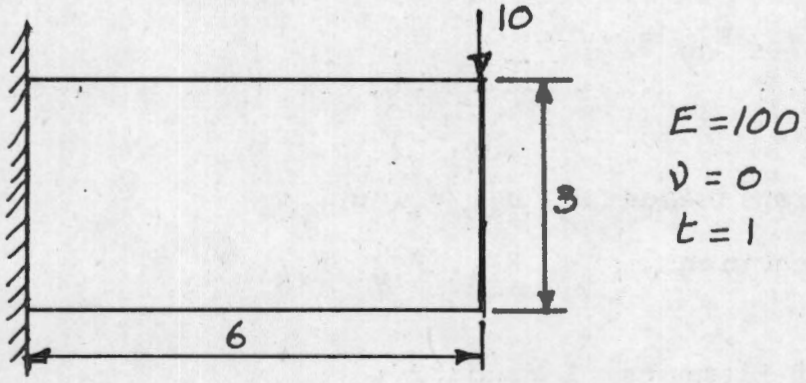
Substituting Eq. (4.39) into Eq. (4.38),

$$\{\varepsilon\}^T [T] [D'] [T]^T \{\varepsilon\} = \{\varepsilon\}^T [D] \{\varepsilon\} \tag{4.41a}$$

$$\text{(or)} \quad [D] = [T] [D'] [T]^T \tag{4.41b}$$

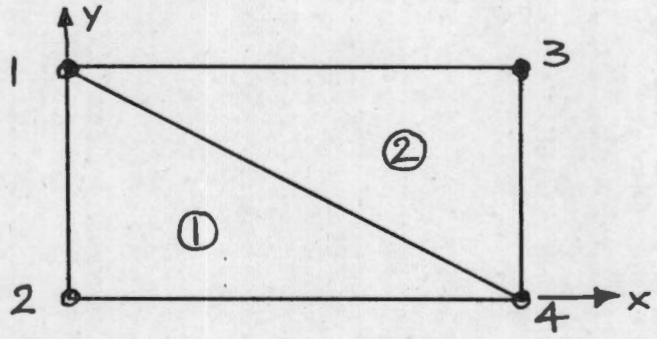
EXAMPLE 4.1

Determine the displacements, strains and stresses for the cantilever beam loaded at the free end as shown in the sketch.



FEM Idealization:

Divide the beam into 2 triangular elements and number the nodes as shown.



DATA:

Nodal co-ordinates	x	y
Node 1	0	3
2	0	0
3	6	3
4	6	0

Nodal connections	Nodes		
Element 1	1	2	4
Element 2	1	4	3

Boundary Conditions

$$u_1 = v_1 = 0$$

$$u_2 = v_2 = 0$$

$$F_{3y} = -10$$

Unkowns:

Displacements: u_3, v_3, u_4, v_4

Reactions: $R_{1x}, R_{1y}, R_{2x}, R_{2y}$

Area of Elements 1 and 2 :

$$2\Delta = \begin{vmatrix} 1 & 0 & 3 \\ 1 & 0 & 0 \\ 1 & 6 & 0 \end{vmatrix} = 18$$

$$\Delta = \frac{18}{2} = 9$$

Element Matrices:

Element 1

1 (0,3) 2 (0,0) 4 (6,0)

$b_1 = y_2 - y_3 = 0$ $c_1 = x_3 - x_2 = 6$

$b_2 = y_3 - y_1 = -3$ $c_2 = x_1 - x_3 = -6$

$b_3 = y_1 - y_2 = 3$ $c_3 = x_2 - x_1 = 0$

$$[B] = \frac{1}{18}\begin{bmatrix} 0 & 0 & -3 & 0 & 3 & 0 \\ 0 & 6 & 0 & -6 & 0 & 0 \\ 6 & 0 & -6 & -3 & 0 & 3 \end{bmatrix}$$

$$[D] = 100 \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & \frac{1}{2} \end{bmatrix}$$

$$[k]_1^e = [B]^T [D] [B] t\Delta$$

$$= 1\times 9 \begin{bmatrix} 0 & 0 & 1/3 \\ 0 & 1/3 & 0 \\ -1/6 & 0 & -1/3 \\ 0 & -1/3 & 1/6 \\ 1/6 & 0 & 0 \\ 0 & 0 & 1/3 \end{bmatrix} \begin{bmatrix} 100 & 0 & 0 \\ 0 & 100 & 0 \\ 0 & 0 & 50 \end{bmatrix} \begin{bmatrix} 0 & 0 & -1/6 & 0 & 1/6 & 0 \\ 0 & 1/3 & 0 & -1/3 & 0 & 0 \\ 1/3 & 0 & -1/3 & 1/6 & 0 & 1/3 \end{bmatrix}$$

$$= \begin{array}{c} \\ 1 \\ \\ 2 \\ \\ 4 \end{array} \begin{array}{c} \begin{array}{cccccc} 1 & & & 2 & & 4 \end{array} \\ \begin{bmatrix} 50 & 0 & -50 & 25 & 0 & 50 \\ 0 & 100 & 0 & -100 & 0 & 0 \\ -50 & 0 & 75 & -25 & -25 & -50 \\ 25 & -100 & -25 & 112.5 & 0 & 25 \\ 0 & 0 & -25 & 0 & 25 & 0 \\ 50 & 0 & -50 & 25 & 0 & 50 \end{bmatrix} \end{array} \begin{array}{c} \\ 1 \\ \\ 2 \\ \\ 4 \end{array}$$

Element 2 $\quad$ 1 (0,3) $\quad$ 4 (6,0) $\quad$ 3 (6,3)

$b_1 = -3 \qquad c_1 = 0$

$b_2 = 0 \qquad c_2 = -6$

$b_3 = 3 \qquad c_3 = 6$

$$[B] = \frac{1}{18}\begin{bmatrix} -3 & 0 & 0 & 0 & 3 & 0 \\ 0 & 0 & 0 & -6 & 0 & 6 \\ 0 & -3 & -6 & 0 & 6 & 3 \end{bmatrix}$$

$$[k^e]_2 = [B]^T [D] [B] t\Delta$$

$$= \begin{array}{c} \begin{matrix} 1 & & & 4 & & 3 \end{matrix} \\ \begin{bmatrix} 25 & 0 & 0 & 0 & -25 & 0 \\ 0 & 12.5 & 25 & -25 & 0 & -12.5 \\ 0 & 25 & 50 & 0 & -50 & -25 \\ 0 & 0 & 0 & 100 & 0 & -100 \\ -25 & -25 & -50 & 0 & 75 & 25 \\ 0 & -12.5 & -25 & -100 & 25 & 112.5 \end{bmatrix} \end{array} \begin{matrix} \\ 1 \\ \\ \\ 4 \\ \\ 3 \end{matrix}$$

Total Stiffness Matrix:

$$[K] = [K]_1 + [K]_2$$

$$= \begin{bmatrix} 75 & 0 & -50 & 25 & -25 & 0 & 0 & 50 \\ & 112.5 & 0 & -100 & 0 & -12.5 & 25 & -25 \\ & & 75 & -25 & 0 & 0 & -25 & -50 \\ & & & 112.5 & 0 & 0 & 0 & 25 \\ & & & & 75 & 25 & -50 & 0 \\ & & & & & 112.5 & -25 & -100 \\ & \text{Symmetric} & & & & & 75 & 0 \\ & & & & & & & 150 \end{bmatrix}$$

Force Displacement Relationship:

$$[K]\ \{u\} = \{P\}$$

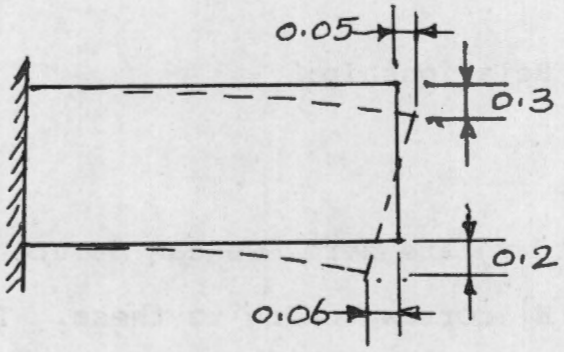
Since u_1, u_2, v_1 and v_2 are zero, we can delete the rows and columns in matrix $[K]$ corresponding to these. Then the stiffness matrix will be only 4×4 and the number of unkowns is 4.

$$\begin{bmatrix} 75 & 25 & -50 & 0 \\ 25 & 112.5 & -25 & -100 \\ -50 & -25 & 75 & 0 \\ 0 & -100 & 0 & 150 \end{bmatrix} \begin{bmatrix} u_3 \\ v_3 \\ u_4 \\ v_4 \end{bmatrix} = \begin{bmatrix} 0 \\ -10 \\ 0 \\ 0 \end{bmatrix}$$

Solving this system of equations,

$$\begin{bmatrix} u_3 \\ v_3 \\ u_4 \\ v_4 \end{bmatrix} = \begin{bmatrix} 0.028 & -0.005 & 0.017 & -0.003 \\ -0.005 & 0.030 & 0.006 & 0.020 \\ 0.017 & 0.006 & 0.027 & 0.004 \\ -0.003 & 0.020 & 0.004 & 0.020 \end{bmatrix} \begin{bmatrix} 0 \\ -10 \\ 0 \\ 0 \end{bmatrix}$$

$$= \begin{bmatrix} 0.05 \\ -0.30 \\ -0.06 \\ -0.20 \end{bmatrix}$$



$$\delta = \frac{PL^3}{3EI} = \frac{10 \times 6^3 \times 12}{3 \times 100 \times 1 \times 3^3} = 3.2 \rightarrow 0.25$$

Stresses: $\{\sigma\} = [D]\ [B]\ \{\delta\}^e$

Element 1

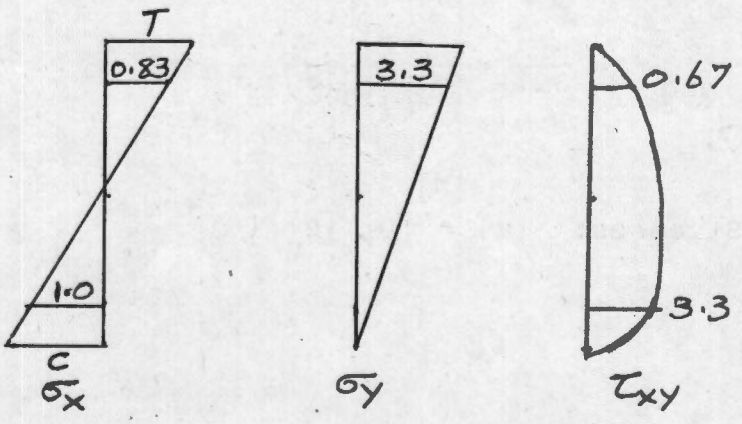
$$\{\sigma\} = \begin{bmatrix}\sigma_x\\ \sigma_y\\ \tau_{xy}\end{bmatrix} = \begin{bmatrix}100 & 0 & 0\\ 0 & 100 & 0\\ 0 & 0 & 50\end{bmatrix}\begin{bmatrix}0 & 0 & -1/6 & 0 & 1/6 & 0\\ 0 & 1/3 & 0 & -1/3 & 0 & 0\\ 1/3 & 0 & -1/3 & 1/6 & 0 & 1/3\end{bmatrix}\begin{bmatrix}0\\ 0\\ 0\\ 0\\ -0.06\\ -0.20\end{bmatrix}$$

$$= \begin{bmatrix}-1.0\\ 0\\ -3.3\end{bmatrix}$$

Element 2

$$\begin{bmatrix}\sigma_x\\ \sigma_y\\ \tau_{xy}\end{bmatrix} = \begin{bmatrix}100 & 0 & 0\\ 0 & 100 & 0\\ 0 & 0 & 50\end{bmatrix}\begin{bmatrix}-1/6 & 0 & 0 & 0 & 1/6 & 0\\ 0 & 0 & 0 & -1/3 & 0 & 1/3\\ 0 & -1/6 & 1/3 & 0 & 1/3 & 1/6\end{bmatrix}\begin{bmatrix}0\\ 0\\ -0.06\\ -0.20\\ 0.05\\ -0.30\end{bmatrix}$$

$$= \begin{bmatrix}0.83\\ -3.3\\ -0.67\end{bmatrix}$$



5. ELEMENT SHAPE FUNCTIONS - ISOPARAMETRIC ELEMENTS

5.1 INTRODUCTION

From the discussion of the previous chapters, it can be seen that the development of the element shape functions plays a main role in the finite element analysis. In the case of one dimensional element (example in Chapter 2) and the triangular element (Chapter 4), the element shape functions have been developed using a polynomial. Thus for any element, the unknown variable, ϕ, can be represented in terms of a polynomial as

$$\phi = [P]\ \{\alpha\} \tag{5.1}$$

where $[P] = [\,1 \quad x \quad y \quad z \quad . \quad . \quad .\,]$ (5.2)

Also $\{\phi\}^e = [c]\ \{\alpha\}$ (5.3)

where
$$[c] = \begin{bmatrix} 1 & x_1 & y_1 & z_1 & . & . & . \\ 1 & x_2 & y_2 & z_2 & . & . & . \\ . & . & . & . & . & . & . \\ . & . & . & . & . & . & . \end{bmatrix} \tag{5.4}$$

Thus, $\{\alpha\} = [c]^{-1}\ \{\phi\}^e$ (5.5)

and $\phi = [P]\ [c]^{-1}\ \{\phi\}^e$

$= [N]\ \{\phi\}^e$ (5.5a)

The method of polynomial representation for obtaining shape functions has certain disadvantages. For example, the matrix [c] may not be inverted and even if it can be inverted, it will involve considerable time for computing the inverse especially in the case of three dimensional complex elements. Therefore, other methods such as the interpolation techniques are employed with much ease in developing the shape functions. Broadly, there are two kinds of interpolations-Lagrangian and Hermitian which are generally used. Or, if one prefers and in certain cases such as shells of revolution with nonsymmetric loading a Fourier series representation can be employed.

5.2 LAGRANGIAN INTERPOLATION

Referring to Fig 5.1, let us say that the variable, ϕ, has certain known values at n points and it is desired to define ϕ at any other point in terms of the values of ϕ at n points. By fitting a curve of polynomial of order (n-1), we can write

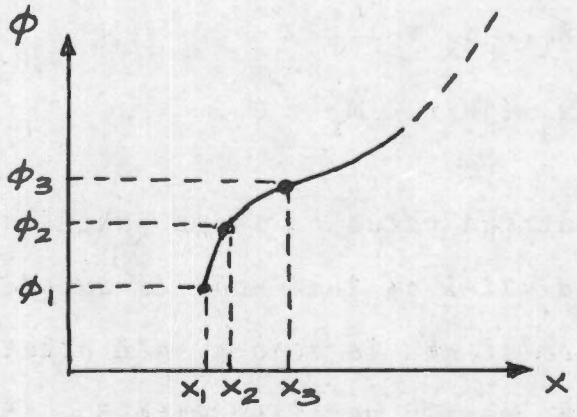


Fig 5.1

$$\phi = N_1 \phi_1 + N_2 \phi_2 + N_3 \phi_3 + \ldots + N_n \phi_n \qquad (5.6)$$

$$N_1 = \frac{(x-x_2)\ (x-x_3)\ \ldots\ (x-x_n)}{(x_1-x_2)(x_1-x_3)\ \ldots\ (x_1-x_n)} \qquad (5.7)$$

$$N_2 = \frac{(x-x_1)\ (x-x_3)\ \ldots\ (x-x_n)}{(x_2-x_1)(x_2-x_3)\ \ldots\ (x_2-x_n)}$$

Similarly $N_3 \ldots N_n$

Eq. (5.7) can be written in general terms as

$$N_i = \frac{\prod_{j=1,\ j\neq i}^{n} (x-x_j)}{\prod_{j=1,\ j\neq i}^{n} (x_i-x_j)} \qquad (5.8)$$

where $\prod$ denotes a product of the indicated binomials over the indicated range j.

An important point can be noted from the study of Eqs. (5.6) to (5.8)

$$\text{For } x = x_i, \quad N_i = 1$$
$$\text{and } x = x_j \ (j\neq i), \quad N_i = 0 \qquad (5.9)$$

Eq. (5.9) states that *the value of shape function at a node is 1 when the co-ordinates of that node is substituted in the interpolating polynomial and is zero at all other nodes.* This statement can be easily verified with Eq. (5.6). For example, if we want the variable ϕ at point 1, then

$$\phi_1 = N_1 \phi_1 + N_2 \phi_2 + \ldots + N_n \phi_n \qquad (5.10)$$

Naturally, $N_1 = 1$ and $N_2 = \ldots N_n = 0$

The shape function given by Eq. (5.8) is for the one dimensional situation shown in Fig 5.1. However, it can be easily extended to either two or three dimensional situations by writing down the product of the suitable functions in x with y or with y and z. Thus in general terms

$$\phi = \sum_{i=1}^{l} \sum_{j=1}^{m} \sum_{k=1}^{n} N_i(x)\, N_j(y)\, N_k(z)\, \phi_{ijk} \qquad (5.11)$$

where

l = number of lines of nodes along x axis
m = number of lines of nodes along y axis
n = number of lines of nodes along z axis

and ϕ_{ijk} is the value of the variable at $x = x_i$, $y = y_i$ and $z = z_i$

5.3 HERMITIAN INTERPOLATION

In some cases, not only the continuity of the distribution of field variable but the continuity of its derivatives should also be satisfied (such as bending of beams and plates). This requirement can be satisfied by the use of Hermitian interpolation.

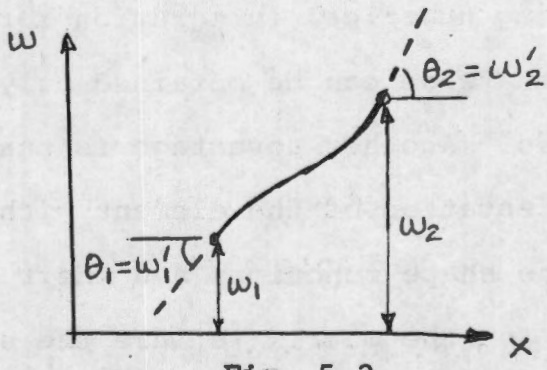


Fig 5.2

Referring to Fig 5.2, the problem prescribes two deflections w_1 and w_2 and two rotations θ_1 and θ_2 at the two ends of, say, a beam. For this case, the interpolation can be accomplished by the use of a cubic polynomial which has four terms.

$$N_i = \alpha_1 + \alpha_2 x + \alpha_3 x^2 + \alpha_4 x^3 \tag{5.12}$$

and

$$w = N_1 w_1 + N_2 w_1' + N_3 w_2 + N_4 w_2' \tag{5.13}$$

5.4 NATURAL (LOCAL) COORDINATE SYSTEM

Contrary to the global co-ordinate system which was used to formulate the triangular element in the previous chapter, the natural co-ordinate system is a local system defined for a particular element. It allows the specification of coordinates of a point in terms of normalized or nondimensionalized co-ordinates which take on values of one or zero at the nodal points.

The major advantage of the natural co-ordinate system is that it not only generalizes the formulation but it also simplifies it in such a way that the stiffness matrix for an element can be generated by using numerical integration. Moreover, it assures invariance which otherwise can be obtained only by using complete polynomials. Another advantage is that irrespective of the size and orientation of the element with respect to the global axes, the shape functions and their derivatives which are required for the matrix [B] are the same for any element. However, there exists a relation between the two

systems and the relationship should be included in calculating the element stiffness matrix. The natural co-ordinate system also allows the easy representation of varying thickness and material properties within an element.

In order to illustrate the use of natural co-ordinates, let us consider first the simple one-dimensional case. Referring to Fig 5.3, the point i is at a distance of ℓ_1 from node 1 and ℓ_2 from node 2.

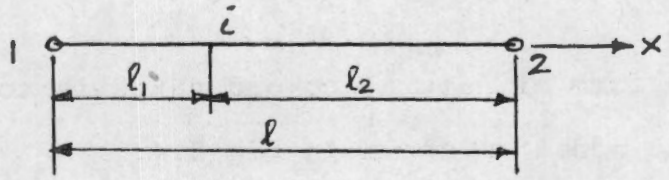


Fig 5.3

Thus $$\ell_1 + \ell_2 = \ell \tag{5.14}$$

or, in terms of normalized co-ordinates defined by

$$L_1 = \frac{\ell_1}{\ell} \text{ and } L_2 = \frac{\ell_2}{\ell} \tag{5.15}$$

$$L_1 + L_2 = 1 \tag{5.16}$$

The co-ordinate x of the point i is then

$$x = L_1 x_1 + L_2 x_2 \tag{5.17}$$

From Eq. (5.17), it can be seen that

$L_1 = 1$ at node 1 and $L_1 = 0$ at node 2

and

$L_2 = 0$ at node 1 and $L_2 = 1$ at node 2

Using Eqs. (5.16) and (5.17), the relationship between natural co-ordinate and the global co-ordinate is

$$\begin{bmatrix} 1 \\ x \end{bmatrix} = \begin{bmatrix} 1 & 1 \\ x_1 & x_2 \end{bmatrix} \begin{bmatrix} L_1 \\ L_2 \end{bmatrix} \tag{5.18}$$

or, by inverting,

$$\begin{bmatrix} L_1 \\ L_2 \end{bmatrix} = \frac{1}{\ell} \begin{bmatrix} x_2 & -1 \\ -x_1 & 1 \end{bmatrix} \begin{bmatrix} 1 \\ x \end{bmatrix} \tag{5.19}$$

An alternate form of natural co-ordinates is to assign unit values at the nodes as shown in Fig 5.4.

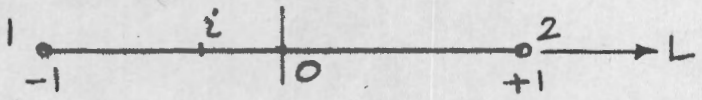


Fig 5.4

In this case, $x = \frac{1}{2}(1-L)\, x_1 + \frac{1}{2}(1+L)\, x_2$ (5.20)

Inverting (5.20), $L = \dfrac{x - (x_1 + x_2)/2}{(x_1 - x_2)/2}$ (5.21)

Thus, $\dfrac{d}{dx} = \dfrac{dL}{dx}\dfrac{d}{dL} = \dfrac{2}{\ell}\dfrac{d}{dL}$ (5.22)

Integration of polynomial terms in the natural co-ordinate system is simpler and is given for instance

$$\int_{\ell} L_1^p\, L_2^q\, d\ell = \frac{p!\, q!}{(p+q+1)!}\, \ell \tag{5.23}$$

5.4.1 TRIANGULAR CO-ORDINATES OR AREA CO-ORDINATES

In the case of triangular elements, the shape functions can be generated conveniently by using the natural co-ordinate system than the cartesian system. The natural co-ordinate system is generally known as area co-ordinates or triangular co-ordinates. The triangular co-ordinate system defines the co-ordinates of a point within an element as follows. Referring to Fig 5.5(a)

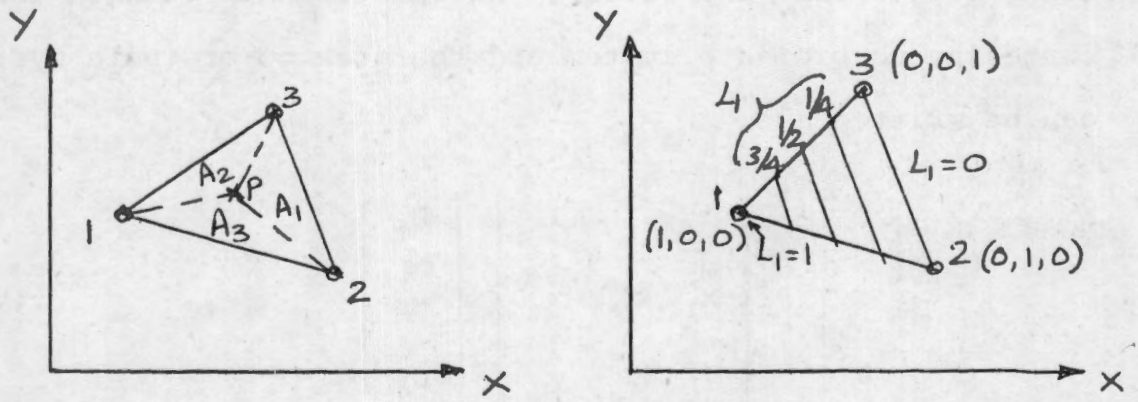


Fig 5.5

the triangle is divided into three parts by joining the point under consideration to the three vertices. Let the areas of these three subdivisions be A_1, A_2 and A_3. If we introduce the local co-ordinate system as L_1, L_2 and L_3, then the relationship between L_1, L_2 and L_3 and A_1, A_2, A_3 can be written as

$$L_1 = \frac{A_1}{A};\ L_2 = \frac{A_2}{A};\ L_3 = \frac{A_3}{A} \qquad (5.24)$$

where A is the area of the triangle 123.

Since $A_1 + A_2 + A_3 = A$ (5.25a)

it follows that $L_1 + L_2 + L_3 = 1$ (5.25b)

From Eq. (5.25b), it can be seen that only two of the three area co-ordinates are independent.

Following Eq. (5.6) and (5.7),

$$x = L_1 x_1 + L_2 x_2 + L_3 x_3$$
$$y = L_1 y_1 + L_2 y_2 + L_3 y_3 \qquad (5.26)$$

Using Eq. (5.25b) and (5.26), the relationship between the cartesian co-ordinate system and the area co-ordinate system can be written as

$$\begin{bmatrix} 1 \\ x \\ y \end{bmatrix} = \begin{bmatrix} 1 & 1 & 1 \\ x_1 & x_2 & x_3 \\ y_1 & y_2 & y_3 \end{bmatrix} \begin{bmatrix} L_1 \\ L_2 \\ L_3 \end{bmatrix} \qquad (5.27)$$

Or, by inversion,

$$\begin{bmatrix} L_1 \\ L_2 \\ L_3 \end{bmatrix} = \frac{1}{2A} \begin{bmatrix} a_1 & b_1 & c_1 \\ a_2 & b_2 & c_2 \\ a_3 & b_3 & c_3 \end{bmatrix} \begin{bmatrix} 1 \\ x \\ y \end{bmatrix} \qquad (5.28)$$

where

$$a_1 = x_2 y_3 - x_3 y_2$$
$$b_1 = y_2 - y_3 \qquad (5.29)$$
$$c_1 = x_3 - x_2$$

The identity of Eq. (5.28) to Eq. (4.6a) and Eq. (5.29) to (4.36) is worth noting.

If the variable ϕ in Eq. (5.10) is considered to be the co-ordinate x or y, then from Eq. (5.26) it follows that

$$N_1 = L_1 \; ; \quad N_2 = L_2 \; ; \quad N_3 = L_3 \tag{5.30}$$

Also from Eq. (5.26),

$$\begin{aligned} &\text{at node 1,} \quad L_1 = 1 \quad \text{and} \quad L_2 = L_3 = 0 \\ &\text{at node 2,} \quad L_2 = 1 \quad \text{and} \quad L_1 = L_3 = 0 \\ &\text{at node 3,} \quad L_3 = 1 \quad \text{and} \quad L_1 = L_2 = 0 \end{aligned} \tag{5.31}$$

Therefore, the mapping of the co-ordinate system can be carried out as shown in Fig 5.5(b) for L_1.

Having obtained the relationship between the cartesian and area co-ordinates, the derivative of any function w.r.t. Cartesian system can be written in terms of local system as

$$\begin{aligned} \frac{\partial}{\partial x} &= \sum_{i=1}^{3} \frac{\partial L_i}{\partial x} \frac{\partial}{\partial L_i} = \sum_{i=1}^{3} \frac{b_i}{2A} \frac{\partial}{\partial L_i} \\ \frac{\partial}{\partial y} &= \sum_{i=1}^{3} \frac{\partial L_i}{\partial y} \frac{\partial}{\partial L_i} = \sum_{i=1}^{3} \frac{c_i}{2A} \frac{\partial}{\partial L_i} \end{aligned} \tag{5.32}$$

The integration of polynomial terms can be carried out using the formula

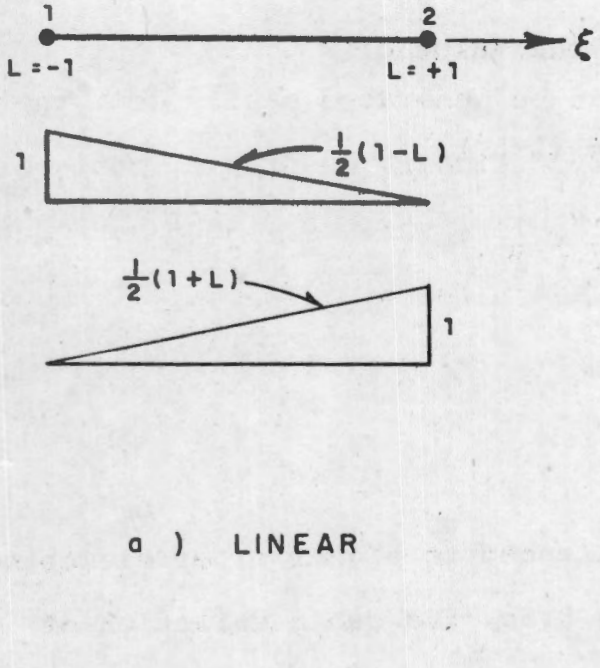
$$\int_A L_1^p \, L_2^q \, L_3^r \, dA = \frac{p!\; q!\; r!}{(p+q+r+2)!} \, 2A \tag{5.33}$$

5.5 INTERPOLATION FUNCTIONS

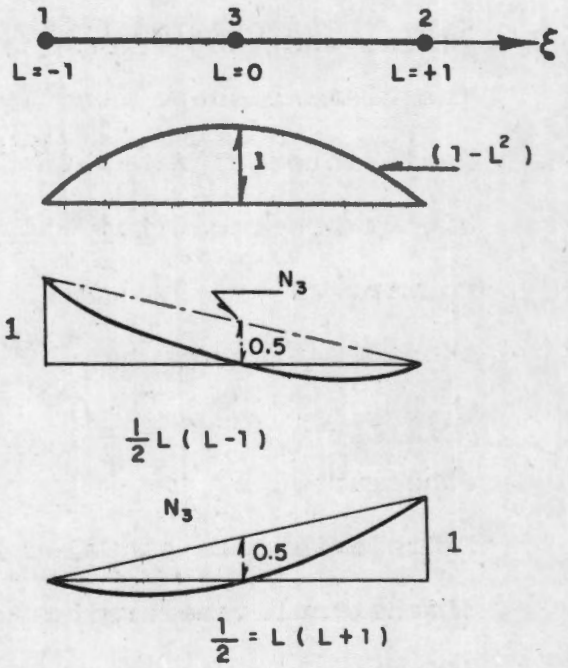
The element shape functions can be generated easily from the basic interpolation functions for linear, quadratic, cubic or higher order variation of the unknown variable. The basic concept of unit value for the shape function at the node under consideration and zero value at all other nodes facilitates this simple generation. The shape functions at a node are constructed by the superposition of component shape functions. In this way the simple interpolation functions for one dimensional case can be easily extended to two and three dimensional cases. Moreover, the use of local or natural co-ordinates is more convenient for the generation of shape functions. The generation of shape functions using the interpolation formulae will be first demonstrated for the case of one dimensional elements.

Referring to Fig 5.6a, since the displacement varies linearly between the two nodes, the interpolation functions are as given in the sketch. If we consider a line element with three nodes, the displacement varies quadratically and hence it will have unit value at the central node as shown in Fig 5.6b and also unit values at the corner nodes. However, it can be seen that if the corner node has unit value, the mid side node has a value of 0.5. This can be eliminated as follows. First, construct the shape function for the mid side node, which is

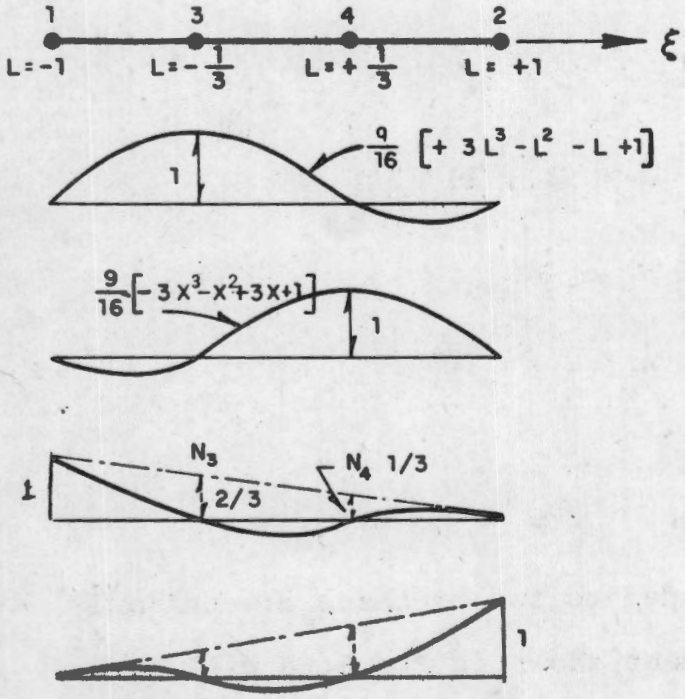
$$N_3 = 1-L^2$$



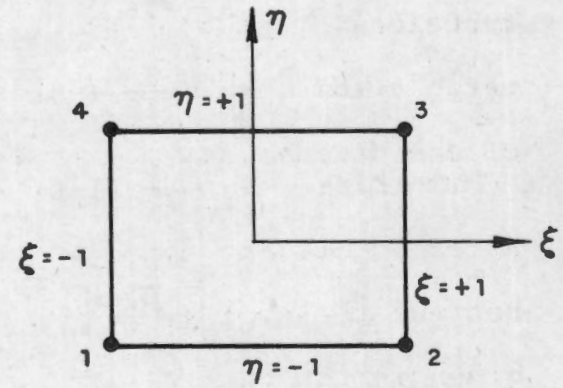
a) LINEAR



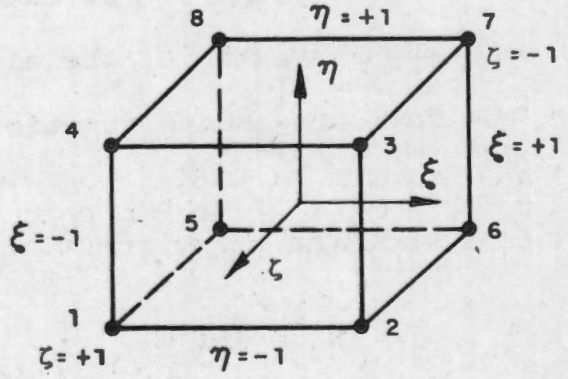
b) QUADRATIC



c) CUBIC



d) 2-D ELEMENT



e) 3-D ELEMENT

Fig 5.6

Then, subtract from N_1, half the value of N_3.

$$N_1 = \frac{1}{2}(1-L) - \frac{1}{2}(1-L^2)$$

$$= \frac{1}{2} - \frac{1}{2}L - \frac{1}{2} + \frac{1}{2}L^2$$

$$= \frac{1}{2}L(L-1)$$

Similarly, $N_2 = \frac{1}{2}L(L+1)$

This principle can also be extended to higher order elements. For example, referring to Fig 5.6c, the cubic variation of displacement is given by

$$N_3 = \frac{9}{16}(3L^3 - L^2 - 3L + 1)$$

and $$N_4 = \frac{9}{16}(-3L^3 - L^2 + 3L + 1)$$

Therefore $$N_1 = \frac{1}{2}(1-L) - \frac{2}{3}N_3 - \frac{1}{3}N_4$$

$$= \frac{1}{16}\left[-9L^3 + 9L^2 + L - 1\right]$$

similarly for N_2.

The same concept can be extended to two or three dimensional cases. For example, the element shown in Fig 5.6d will have the following shape functions.

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

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

In general $N_i = \frac{1}{4} (1+\xi\ \xi_i)\ (1+\eta\ \eta_i)$

where $\xi_i = \mp 1$ and $\eta_i = \mp 1$

In the case of three dimensional element shown in Fig 5.6e,

$$N_2 = \left[\frac{1}{2} (1+\xi)\right]\left[\frac{1}{2} (1-\eta)\right]\left[\frac{1}{2} (1+\zeta)\right]$$

$$= \frac{1}{8} (1+\xi)\ (1-\eta)\ (1+\zeta)$$

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

where $\xi_i = \mp 1$, $\eta_i = \mp 1$ and $\zeta_i = \mp 1$

The shape functions for higher order elements can be generated in a similar manner to that described for the one dimensional element except using the product of appropriate components selected from the one dimensional case. The shape functions for some of the commonly used elements are given in Table 5.1

In the case of triangular elements, the shape functions can be developed using the area co-ordinates. The shape functions at a node is generated by the product of the appropriate component shape functions obtained for each direction using Lagrangian interpolation.

For the constant strain triangle with 3 corner nodes, it can be easily written from Eq. (5.30) as

$$N_1 = L_1, \quad N_2 = L_2, \quad N_3 = L_3 \qquad (5.34)$$

so that $\phi = N_1 \phi_1 + N_2 \phi_2 + N_3 \phi_3$ (5.35)

In the case of higher order elements, the shape functions can be generated in such a way that 1) the polynomial is of the particular degree, 2) it includes constant strain and rigid body movement terms and 3) has equal number of terms as that of its order.

For example, in the case of linear strain triangle with 3 corner nodes and 3 midside nodes as shown in Fig 5.7, the interpolation

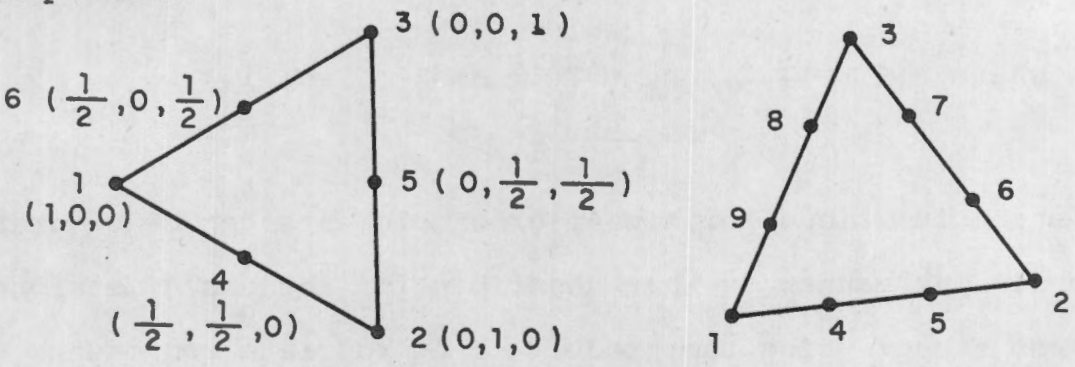


Fig 5.7

function should be quadratic and should have two terms.

It can be written for a polynomial of n degree, the corner nodes will have shape functions of the form

$$\frac{1}{(n-1)!} L_i (nL_i - 1)(nL_i - 2) \ldots (nL_i - (n-1)) \quad (5.36)$$

and the interior nodes along a side will have shape functions of the form

$$\frac{n^2}{(n-1)!} L_i L_j (nL_i - 1)(nL_i - 2) \ldots (nL_i - (n-2)) \quad (5.37)$$

For node 1 , $N_1 = L_1 \ (2L_1 - 1)$

For node 4 , $N_4 = \frac{2^2}{1!} L_1 \ L_2 = 4L_1 \ L_2$

In the case of triangle with cubic variation of displacement, (n=3).

For corner node, $N_1 = \frac{1}{2!} L_1 \ (3L_1-1) \ (3L_1-2)$

For interior nodes, $N_4 = \frac{3^2}{2!} L_1 \ L_2 \ (3L_1-1)$

$= \frac{9}{2} L_1 \ L_2 \ (3L_1- 1)$

For internal node at the centroid, $N_{10} = 27L_1 \ L_2 \ L_3$.

TABLE 5.1 TYPICAL SHAPE FUNCTIONS

ELEMENT	LINEAR	QUADRATIC	CUBIC
Triangle	$N_i = L_i$	$N_i = L_i\,(2L_i - 1)$ for corner nodes a) $N_i = 4L_j\,L_k$	$N_i = \frac{1}{2}\,L_i\,(3L_i - 1)(3L_i - 2)$ for corner nodes b) $N_i = \frac{9}{2}\,L_j\,L_k\,(3L_j - 1)$ for node at 1/3 c) $N_i = 27L_j\,L_k\,L_\ell$ for internal node
Quadrilateral	$N_i = \frac{1}{4}\,(1+\xi\,\xi_i)\times(1+\eta\,\eta_i)$ $\xi_i = \pm 1,\ \eta_i = \pm 1$	$N_i = \frac{1}{4}\,(1+\xi\,\xi_i)(1+\eta\,\eta_i)\times(\xi\,\xi_i + \eta\,\eta_i - 1)$ for corner nodes $N_i = \frac{1}{2}\,(1-\xi^2)(1+\eta\,\eta_i)$ for $\xi_i = 0$ $N_i = \frac{1}{2}\,(1+\xi\,\xi_i)(1-\eta^2)$ for $\eta_i = 0$	$N_i = \frac{1}{32}(1+\xi\,\xi_i)(1+\eta\,\eta_i)\times[-10 + 9(\xi^2+\eta^2)]$ for corner nodes $N_i = \frac{9}{32}(1+\xi\,\xi_i)(1-\eta^2)\times(1+9\eta\,\eta_i)$ for $\xi_i = \pm 1,\ \eta_i = \pm\frac{1}{3}$
Rectangular Prism.	$N_i = \frac{1}{8}(1+\xi\,\xi_i)(1+\eta\,\eta_i)(1+\zeta\,\zeta_i)$ $\xi_i = \pm 1,\ \eta_i = \pm 1,\ \zeta_i = \pm 1$	$N_i = \frac{1}{8}\,(1+\xi\,\xi_i)(1+\eta\,\eta_i)\times(1+\zeta\,\zeta_i)(\xi\,\xi_i+\eta\,\eta_i+\zeta\,\zeta_i-2)$ for corner nodes $N_i = \frac{1}{4}(1-\xi^2)(1+\eta\,\eta_i)(1+\zeta\,\zeta_i)$ for $\xi_i=0,\ \eta_i=\pm 1,\ \zeta_i=\pm 1$	$N_i = \frac{1}{64}\,(1+\xi\,\xi_i)(1+\eta\,\eta_i)\times(1+\zeta\,\zeta_i)[9(\xi^2+\eta^2+\zeta^2)-19]$ for corner nodes $N_i = \frac{9}{64}\,(1-\xi^2)(1+9\xi\,\xi_i)\times(1+\eta\,\eta_i)\,(1+\zeta\,\zeta_i)$ for $\xi_i = \pm\frac{1}{3},\ \eta_i = \pm 1,\ \zeta_i = \pm 1$

5.6 ISOPARAMETRIC ELEMENTS

The need for modelling curved boundaries combined with the idea of natural or local co-ordinates and the applicability of numerical integration has led to the development of *isoparametric elements*. The local co-ordinate system for curved isoparametric elements is curvilinear. Since they are parametrically equivalent to their rectilinear counterparts, they are called isoparametric elements. Similar to the rectilinear elements, curved isoparametric elements also satisfy the compatibility at the interface between elements, continuity within the element and the constant strain as well as rigid body movement criteria.

As discussed in the previous sections, any field variable, whether it be co-ordinates, displacements, material properties, temperature etc. at a point within an element can be uniquely defined in terms of the nodal values by the shape functions. Therefore, if the same shape function (of equal order) is used to define the variable, then such a representation is known as *isoparametric*. If the geometric representation is of higher order than the field variable, it is known as *superparametric*. On the other hand, if the geometric representation is of lower order than that of the field variable, it is termed *subparametric*. Since the isoparametric elements are the only ones which are widely used, the present discussion will be limited to them.

As in the case of their parent elements (rectilinear elements), in isoparametric elements also the value of the shape function will be unity at a particular node under consideration and zero at

other nodes. It is important to take care not to distort the curved edge unreasonably. Otherwise two shape functions may take unit value at the same node.

The basic rectilinear elements in one, two and three dimensions can be mapped into distorted curvilinear forms and an one-to-one correspondence between the cartesian and the curvilinear co-ordinates can be established. Then, the finite element equations can be developed in the local curvilinear system and then transformed into global system.

Let the field variable ϕ be defined in terms of the shape function as

$$\phi = [\, N\ (\xi,\eta,\zeta)\ ]\ \{\phi\}^e \tag{5.38}$$

where the shape function is defined in the local curvilinear system.

For representing co-ordinates,

$$\begin{aligned} x &= [\, N\ (\xi,\eta,\zeta)\ ]\ \{x\}^e \\ y &= [\, N\ (\xi,\eta,\zeta)\ ]\ \{y\}^e \\ z &= [\, N\ (\xi,\eta,\zeta)\ ]\ \{z\}^e \end{aligned} \tag{5.39}$$

For representing displacements,

$$\begin{aligned} u &= [\, N\ (\xi,\eta,\zeta)\ ]\ \{u\}^e \\ v &= [\, N\ (\xi,\eta,\zeta)\ ]\ \{v\}^e \\ w &= [\, N\ (\xi,\eta,\zeta)\ ]\ \{w\}^e \end{aligned} \tag{5.40}$$

The shape functions in Eqs. (5.39) and (5.40) will be identical to those developed in Sections 5.4 and 5.5 for one, two and three dimensional elements (linear, quadratic and cubic).

5.6.1 CONVERGENCE CRITERIA

Similar to the parent elements, the isoparametric elements also have to satisfy the following criteria.

1. If two adjacent elements are generated from "parent" elements in which the shape functions satisfy continuity requirements, then the distorted elements will also satisfy the same continuity condition.
2. If the shape functions are such that the continuity of the field variable is preserved in the parent element, then the same condition is satisfied in the distorted element.

   The second criteria needs no explanation. Considering the first criteria, it can be said that since the field variable, for example, displacements along any edge is uniquely defined by the values at the nodes along that edge only, the interelement compatibility is preserved. It can be readily seen that the shape functions at the common edge of two adjacent elements are identical, thus defining the element shapes before deformation and preserving the continuity of the displacement after deformation such that the common boundaries of two adjacent elements match exactly at all times.
3. The constant strain criteria and the rigid body motion are also satisfied in the isoparametric elements, if the sum of the shape functions is equal to unity.

   This can be proved as follows. Let the field variable be defined uniquely as

$$\phi = [N] \{\phi\}^e = \Sigma N_i \phi_i \qquad (5.41a)$$

Also $$\phi = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 z \tag{5.41b}$$

At a particular node, i, Eq. (5.41b) can be written as

$$\phi_i = \alpha_1 + \alpha_2 x_i + \alpha_3 y_i + \alpha_4 z_i \tag{5.42}$$

Thus, $$\Sigma N_i \phi_i = \alpha_1 \Sigma N_i + \alpha_2 \Sigma N_i x_i + \alpha_3 \Sigma N_i y_i + \Sigma N_i z_i \tag{5.43}$$

From Eq. (5.39) $$x = \Sigma N_i x_i$$

$$y = \Sigma N_i y_i$$

$$z = \Sigma N_i z_i$$

Therefore, $$\Sigma N_i \phi_i = \alpha_1 \Sigma N_i + \alpha_2 x + \alpha_3 y + \alpha_4 z \tag{5.44}$$

Comparing Eq. (5.44) with Eq. (5.41), it can be seen that this identity will be satisfied only if

$$\Sigma N_i = 1 \tag{5.45}$$

Since the polynomial assumed to define the variable consists of a constant term and the lower order terms specified continuously it is simple to verify that rigid body motions and constant strain criteria are satisfied as in the case of the parent elements.

6. ELEMENT EQUATIONS - LINEAR ELASTICITY

6.1 INTRODUCTION

In this chapter, the finite element equations such as the stiffness matrix and the mass or load matrix will be developed for a single element on the basis of shape functions described in Chapter 5.

For the present discussion, if we consider only analysis of problems involving displacements, strains and stresses, then the relationship between strain and displacement is linear and the relationship between stress and strain is elastic based on Hooke's law.

For generating the stiffness matrix of an element which is given by

$$[k^e] = \int [B]^T [D] [B] \, dv \qquad (6.1)$$

we have to form the matrices $[D]$ and $[B]$.
The matrix $[D]$ can be formed either for 2D, 3D or axisymmetric cases based on isotropic or anisotropic material behaviour, as given in Chapter 4. The matrix $[B]$ on the basis of infinitesimal strain theory is given by Eq. (4.21). If the local co-ordinate system is adopted as in the case of isoparametric elements, then the shape functions as in Eq. (5.38) will be represented in the curvilinear system, ξ,η,ζ. Thus it is necessary 1) to transform the matrix $[B]$ from local to global

system, and 2) to express the integration over the volume of an element in terms of local co-ordinates with the limits of integration being -1 to +1 for the cases discussed in Chapter 5.

6.2 TRANSFORMATION IN LOCAL CO-ORDINATES

The strain-displacement matrix is given by

$$[B_i] = \begin{bmatrix} \frac{\partial N_i}{\partial x} & 0 & 0 \\ 0 & \frac{\partial N_i}{\partial y} & 0 \\ 0 & 0 & \frac{\partial N_i}{\partial z} \\ \frac{\partial N_i}{\partial y} & \frac{\partial N_i}{\partial x} & 0 \\ 0 & \frac{\partial N_i}{\partial z} & \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} & 0 & \frac{\partial N_i}{\partial x} \end{bmatrix} \tag{6.2}$$

If we represent the shape function in local co-ordinates such that

$$\phi = [\, N\ (\xi, \eta, \zeta)\ ]\ \{\phi\}^e \tag{6.3}$$

then the derivatives $\frac{\partial N}{\partial \xi}$, $\frac{\partial N}{\partial \eta}$ and $\frac{\partial N}{\partial \zeta}$ can be written by the rules of partial differentiation as

$$\frac{\partial N_i}{\partial \xi} = \frac{\partial N_i}{\partial x}\frac{\partial x}{\partial \xi} + \frac{\partial N_i}{\partial y}\frac{\partial y}{\partial \xi} + \frac{\partial N_i}{\partial z}\frac{\partial z}{\partial \xi} \tag{6.4}$$

Similarly $\frac{\partial N_i}{\partial \eta}$ and $\frac{\partial N_i}{\partial \zeta}$. Therefore, in matrix form

$$\begin{bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \\ \frac{\partial N_i}{\partial \zeta} \end{bmatrix} = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & \frac{\partial z}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & \frac{\partial z}{\partial \eta} \\ \frac{\partial x}{\partial \zeta} & \frac{\partial y}{\partial \zeta} & \frac{\partial z}{\partial \zeta} \end{bmatrix} \begin{bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} \end{bmatrix} = [J] \begin{bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} \end{bmatrix} \qquad (6.5)$$

where $[J]$ is the Jacobian Matrix.

Eq. (6.5) can be written in order to relate the global derivatives of N w.r.t. local as

$$\begin{bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} \end{bmatrix} = [J]^{-1} \begin{bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \\ \frac{\partial N_i}{\partial \zeta} \end{bmatrix} \qquad (6.6)$$

In fact, $$[J] = \begin{bmatrix} \frac{\partial N_1}{\partial \xi} & \frac{\partial N_2}{\partial \xi} & \cdots \\ \frac{\partial N_1}{\partial \eta} & \frac{\partial N_2}{\partial \eta} & \cdots \\ \frac{\partial N_1}{\partial \zeta} & \frac{\partial N_2}{\partial \zeta} & \cdots \end{bmatrix} \begin{bmatrix} x_1 & y_1 & z_1 \\ x_2 & y_2 & z_2 \\ x_3 & y_3 & z_3 \\ \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot \end{bmatrix} \qquad (6.7)$$

$$= [H]\ [\phi]^e$$

Since $x = [N]\ \{x\}^e$, $y = [N]\ \{y\}^e$ and $z = [N]\ \{z\}^e$

and $[N] = [\ N\ (\xi, \eta, \zeta)\ ]$

Therefore, $$[B] = [J]^{-1}\ [H] \qquad (6.8)$$

Thus, $$\{\varepsilon\} = [B]\ \{\delta\}^e = [J]^{-1}\ [H]\ \{\delta\}^e \qquad (6.9)$$

From Eq. (6.8), it can be seen that the Jacobian has to be inverted to obtain the matrix [B]. The Jacobian, in turn, is defined in terms of the co-ordinates of the nodes. Therefore, it is sensitive to certain types of distorted shapes of curved elements. It is important to check on the value of Jacobian; if it is zero or negative, it could be due to the unreasonable distortion.

The integration of the stiffness matrix over the volume involves

$$\int dx\, dy\, dz \tag{6.10a}$$

which is equivalent in local system to

$$\int \det [J]\, d\xi\ d\eta\ d\zeta \tag{6.10b}$$

Since the local co-ordinate system is expressed with limits -1 and +1,

$$[k^e] = \int_{-1}^{+1} \int_{-1}^{+1} \int_{-1}^{+1} [B]^T [D] [B] \det [J]\, d\xi\, d\eta\, d\zeta \tag{6.11}$$

and $$\det [J] = \frac{\partial(x,y,z)}{\partial(\xi,\eta,\zeta)} \tag{6.12}$$

The limits of integration as given in Eq. (6.11) are simple. However, the explicit form of Jacobian matrix and hence the evaluation of stiffness matrix is not. Therefore, numerical integration which is approximate has to be resorted to rather than the exact integration.

6.3 NUMERICAL INTEGRATION

There are a number of methods for the numerical integration of a function. In this section, the discussion will be limited to Gauss quadrature.

If the integral of the following function is to be evaluated,

$$I = \int_{-1}^{+1} y \, dx \tag{6.13}$$

we can evaluate the function at several sampling points, multiply each value of "y_i" by the appropriate weighting coefficient "w_i" and add the products. For example, considering only one point, the aproximate integral will be

$$I = 2y_1 \tag{6.14}$$

where y_1 is the value of the function at the midpoint of the interval.

From Eq. (6.14) it can be noted that the approximate integral of a function can be obtained by evaluating the function at several sampling points, multiplying such values with appropriate weighting coefficients and then adding them up. Thus for a general case, Eq. (6.14) can be written as

$$I = \int_{-1}^{+1} f(x) \, dx \approx \sum_{i=1}^{n} W_i \, f(x_i) \tag{6.15}$$

In Gauss quadrature, the sampling points are located such that the desired accuracy is obtained for a particular set of sampling points. The sampling points are located symmetrically with respect to the centre of the inverval.

Eq. (6.15) can be written for two and three dimensions, using local co-ordinates as follows

$$I = \int_{-1}^{+1} \int_{-1}^{+1} f(\xi,\eta)\, d\xi\, d\eta$$

$$= \sum_{i=1}^{m} \sum_{j=1}^{n} W_i\, W_j\, f(\xi_i, \eta_i) \qquad (6.16)$$

$$I = \int_{-1}^{+1} \int_{-1}^{+1} \int_{-1}^{+1} f(\xi,\eta,\zeta)\, d\xi\, d\eta\, d\zeta$$

$$= \sum_{i=1}^{\ell} \sum_{j=1}^{m} \sum_{k=1}^{n} W_i\, W_j\, W_k\, f(\xi_i, \eta_i, \zeta_i) \qquad (6.17)$$

where ℓ, m, n are number of sampling points in each direction ξ, n, ζ.

For triangular elements, the area co-ordinates are used in the numerical integration

$$I = \int_0^1 \int_0^{1-L_1} f(L_1\ L_2\ L_3)\, dL_1\, dL_2 \qquad (6.18)$$

TABLE 6.1

Gauss quadrature formula $\int_{-1}^{+1} f(x)\,dx = \sum_{i=1}^{n} W_i\, f(a_j)$

	$\pm a_j$	W_i
n = 2	0.57735027	1.0
n = 3	0.77459667	0.55555556
	0.	0.88888889
n = 4	0.86113631	0.34785485
	0.33998104	0.65214515

TABLE 6.2

Numerical Integration for Triangles

Type of element	Points	Triangular co-ordinates	Weighting coefficients, $2W_k$
linear			
	a	1/3, 1/3, 1/3	1
quadratic			
	a	1/2, 1/2, 0	1/3
	b	0 , 1/2, 1/2	1/3
	c	1/2, 0 , 1/2	1/3
cubic			
	a	1/3, 1/3, 1/3	27/60
	b	1/2, 1/2, 0	8/60
	c	0 , 1/2, 1/2	
	d	1/2, 0 , 1/2	
	e	1, 0, 0	3/60
	f	0, 1, 0	
	g	0, 0, 1	

$$\text{or} \quad I = \sum_{i=1}^{m} W_i \, f(L_1 \; L_2 \; L_3) \tag{6.19}$$

The co-ordinates of the sampling points and their weighting coefficients for Gauss Quadrature are given in Table 6.1 whereas the same, for triangular elements, are given in Table 6.2.

6.3.1 NUMBER OF GAUSS POINTS FOR DESIRED ACCURACY

The numerical integration using Gauss Quadrature involves evaluating the function at n Gauss points. Thus the effort for two dimensional problems is roughly proportional to n^2 whereas it is n^3 for three dimensional situations. Therefore it is important to determine an adequate minimum number for desired accuracy. Since the integration of element equation affects the convergence, it can be said that convergence will occur in elastic displacement analysis if the integration is sufficient to evaluate the volume of the element exactly. The evaluation of the volume involves the calculation of determinant of Jacobian. Therefore, the necessary number of Gauss points can be determined by examining the Jacobian.

For a plane linear element, the determinant of [J] is linear and hence only one point is adequate, whereas 2 x 2 points are necessary for quadratic element and 3 x 3 points for cubic element and similarly for three dimensional elements.

In choosing the number of points, it is necessary to use experience since lower order integration will be adequate if

a fine mesh is used and higher order is necessary if a coarse mesh is used. However, the number should never be less than the minimum necessary for convergence.

As mentioned earlier, the finite element displacement formulation results in increased stiffness matrix of an element. Therefore, if the number of points is reduced, then the numerical integration reduces the element stiffness. This may be preferred in many instances.

BIBLIOTECA DE LA UNIVERSIDAD NACIONAL DE MEXICO DEPA

6.4 EQUIVALENT NODAL FORCES

When external forces are concentrated at points, they can be applied directly at nodes. On the other hand, if there are distributed loads either over a surface of an element or the volume of an element, then the forces which are statically equivalent to the distributed loading have to be applied at the corresponding nodes of an element. These equivalent nodal forces can be calculated either directly (which is usually by intuition and is approximate), or by the variational approach which is exact since it conforms with the formulation of the stiffness matrix for the element. The former is generally known as lumped loads whereas the latter is called the consistent loads.

For calculating the consistent loads, Eqs. (4.13) and (4.15) are used respectively for surface pressures and distributed body forces.

$$\{F\}^e = \int_S [N]^T \{T\}\, dS \qquad (4.13)$$

$$\text{and} \quad \{F\}^e = -\int_V [N]^T \{p\}\, dV \qquad (4.15)$$

In the case of isoparametric elements, these loads are calculated at the same time as the element stiffness matrix is computed, using numerical integration. Fig (6.1) shows the allocation of equivalent forces at the nodes of rectilinear elements due to surface load for 2D and 3D as well as the body force.

Even though the final result may be nearly the same in the limit whether lumped loads or consistent loads are used, in the representation of curvilinear elements, a proper allocation should be made due to a more complex distribution of loading.

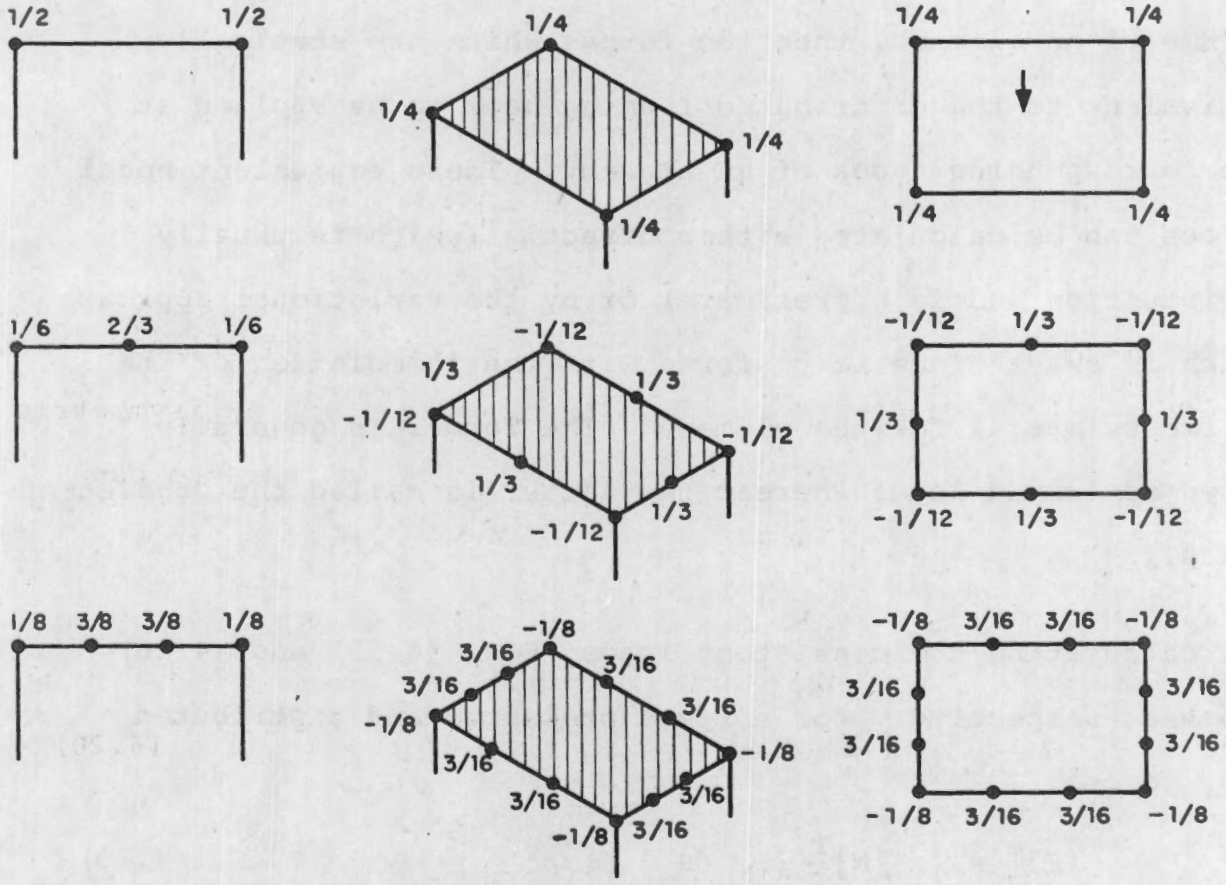


Fig 6.1

6.5 AXISYMMETRIC PROBLEMS WITH NONAXISYMMETRIC LOADS

In practice, situations quite frequently arise where structures which are geometrically axisymmetric are subjected to nonaxisymmetric loading. For example, laterally loaded pile in soil medium, circular chimneys acted upon by wind loads and tanks and cylindrical structures subjected to earthquake. In the analysis of these structures, the use of Fourier series expansion is made and the three dimensional problem is still reduced to a two dimensional one. However, regarding the displacements not only the radial and axial components but the circumferential component should also be considered.

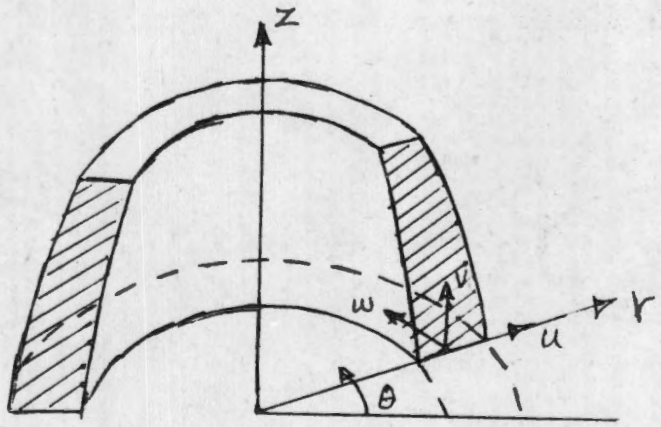


Fig 6.2

For the sake of simplicity, consider the components of load which are symmetric about $\theta = 0$ axis and which are antisymmetric separately. The forces per unit length of circumference can be specified as

$$R = \sum_{n=1}^{N} \bar{R}^n \cos n\theta$$

$$Z = \sum_{n=1}^{N} \bar{Z}^n \cos n\theta \qquad (6.20)$$

$$T = \sum_{n=1}^{N} \bar{T}^n \sin n\theta$$

where

n = is the order of harmonic

N = is the number of harmonics

Eq. (6.20) is given for symmetric loads. For antisymmetric loads, sines and cosines are interchanged, as can be noted from the expression for T whose direction changes for $\theta > \pi$ as shown in Fig 6.3.

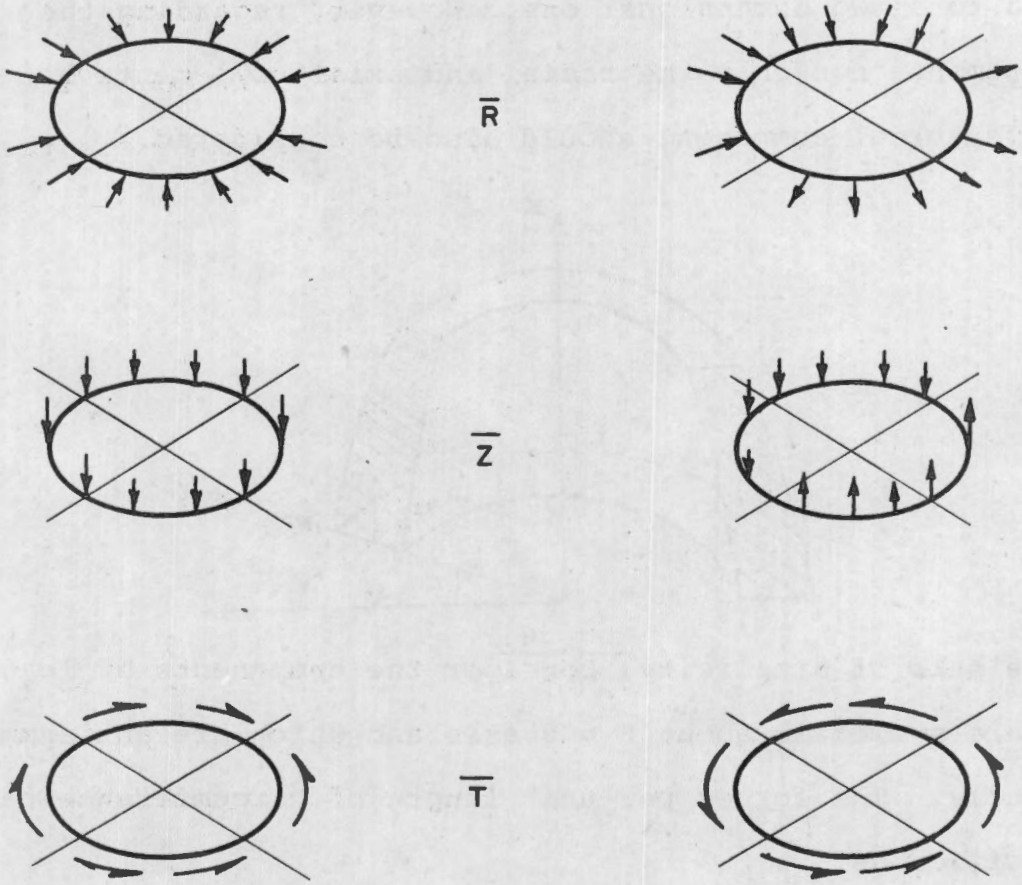


Fig 6.3

Corresponding to the load components, the displacement components can be written using the shape functions developed in two dimensions (r, z).

$$u_n = \begin{bmatrix} N_1 & N_2 & . & . & . \end{bmatrix} \cos n\theta \{u_n\}^e$$

$$v_n = \begin{bmatrix} N_1 & N_2 & . & . & . \end{bmatrix} \cos n\theta \{v_n\}^e \qquad (6.21)$$

$$w_n = \begin{bmatrix} N_1 & N_2 & . & . & . \end{bmatrix} \sin n\theta \{w_n\}^e$$

The strain-displacement relationship now consists of six terms due to the circumferential displacement.

$$\{\varepsilon\} = \begin{bmatrix} \varepsilon_r \\ \varepsilon_z \\ \varepsilon_\theta \\ \gamma_{rz} \\ \gamma_{r\theta} \\ \gamma_{z\theta} \end{bmatrix} = \begin{bmatrix} \partial u/\partial r \\ \partial v/\partial z \\ \frac{u}{r} + \frac{1}{r}\frac{\partial w}{\partial \theta} \\ \frac{\partial u}{\partial z} + \frac{\partial v}{\partial r} \\ \frac{1}{r}\frac{\partial u}{\partial \theta} + \frac{\partial w}{\partial r} - \frac{w}{r} \\ \frac{1}{r}\frac{\partial v}{\partial \theta} + \frac{\partial w}{\partial z} \end{bmatrix} \qquad (6.22)$$

For each harmonic, the strain components and hence the stiffness matrix have to be evaluated since uncoupling occurs between the nodes. Thus, writing the matrix [B] for a single harmonic,

$$[B_i]_n = \begin{bmatrix} \frac{\partial N_i}{\partial r}\cos n\theta & 0 & 0 \\ 0 & \frac{\partial N_i}{\partial z}\cos n\theta & 0 \\ \frac{N_i}{r}\cos n\theta & 0 & -\frac{nN_i}{r}\cos n\theta \\ \frac{\partial N_i}{\partial z}\cos n\theta & \frac{\partial N_i}{\partial r}\cos n\theta & 0 \\ -\frac{nN_i}{r}\sin n\theta & 0 & \left[\frac{\partial N_i}{\partial r} - \frac{N_i}{r}\right]\sin n\theta \\ 0 & -\frac{nN_i}{r}\sin n\theta & \frac{\partial N_i}{\partial z}\sin n\theta \end{bmatrix}$$

i=1 ... no. of nodes

(6.23)

Now, the stiffness matrix is given by

$$[k]_n^e = \int\int\int [B]_n^T [D] [B]_n \, r \, d\theta \, dr \, dz \qquad (6.24)$$

In a similar manner, the stiffness matrix for antisymmetric loading can be evaluated.

The load vector in each harmonic for the symmetric case will be

$$\{F\}_n^e = \int_0^{2\P} \begin{bmatrix} \overline{R}_n \cos^2 n\theta \\ \overline{Z}_n \cos^2 n\theta \\ \overline{T}_n \sin^2 n\theta \end{bmatrix} d\theta \qquad (6.25a)$$

For n = 0, $= 2\pi \begin{bmatrix} \bar{R}_n \\ \bar{Z}_n \\ 0 \end{bmatrix}$ (6.25b)

For n=1, 2 . . . $= \pi \begin{bmatrix} \bar{R}_n \\ \bar{Z}_n \\ \bar{T}_n \end{bmatrix}$ (6.25c)

For the antisymmetric loading,

$$\{F\}_n^e = 2\pi \begin{bmatrix} 0 \\ 0 \\ \bar{T}_n \end{bmatrix} \quad \text{for } n = 0 \tag{6.25d}$$

$$= \pi \begin{bmatrix} \bar{R}_n \\ \bar{Z}_n \\ \bar{T}_n \end{bmatrix} \quad \text{for } n = 1,2 \ldots \tag{6.25e}$$

It can be noted that for n = 0, the problem reduces to only two variables, and for the symmetric loading it reduces to axisymmetric case. Further, it can also be noted that the circumferential co-ordinate is eliminated from the integration because the only non zero terms are $\cos^2 n\theta$ and $\sin^2 n\theta$ and hence

$$\int_0^{2\pi} \cos^2 n\theta \, d\theta = \int_0^{2\pi} \sin^2 n\theta \, d\theta = \pi \quad \text{for } m = n \neq 0$$

From the foregoing discussion, it can be seen that the axisymmetric problem with nonaxisymmetric loading is divided into two separate analyses involving symmetric and antisymmetric modes for each harmonic (Fig 6.4).

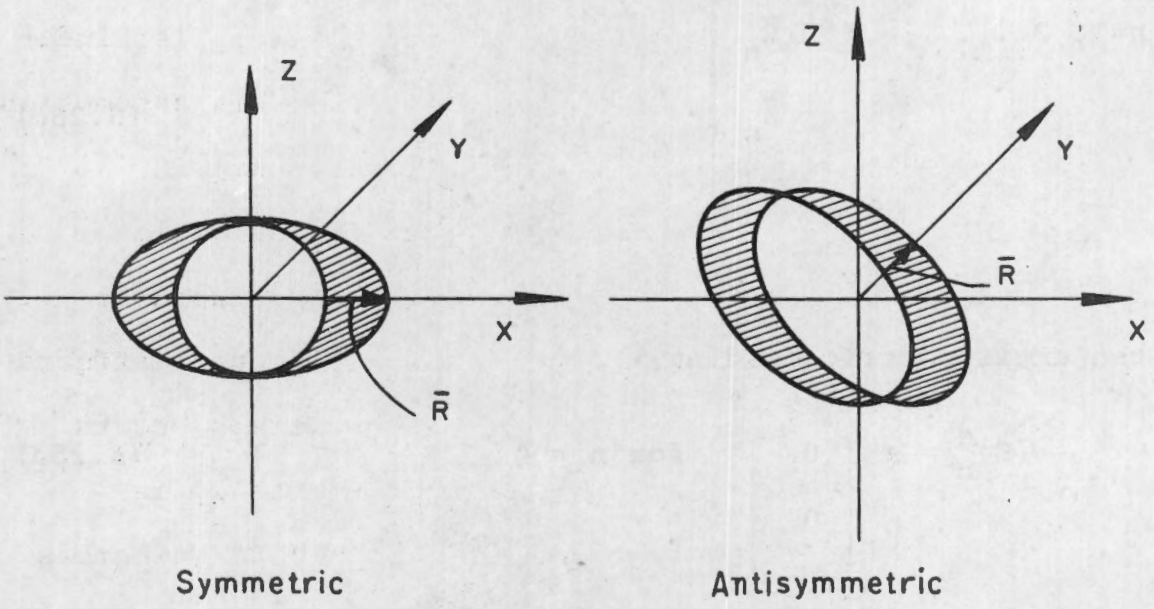


Fig 6.4

The final result for the problem will be the sum of the respective solutions obtained separately. Certain remarks regarding the boundary condition are worth making. When n=1, three fixed boundary conditions are required, for $n > 1$, only axial displacement is to be zero for $n = 1$ and for $n = 0$, axial and tangential components are to be zero.

7. NONLINEAR ANALYSIS - SOLUTION TECHNIQUES

7.1 INTRODUCTION

In Chapter 6, the element equations were developed on the assumption that the strain-displacement relationship is linear and the stress-strain relationship is elastic. This assumption is, in fact, not true for most of the materials used in engineering. Either the material behaviour is inelastic introducing plastic and/or creep strains or the structural component and the loading as well as the environmental effects are such that large displacement and/or large strain occurs. The former case, where the stress-strain relationship is inelastic, is generally known as *material nonlinearity* whereas the latter case in which the strain-displacement relationship is nonlinear is known as *geometric nonlinearity*.

In order to solve the nonlinear problems using FEM, two broad classes of solution techniques are available. In one class, when the material behaviour in an element departs from the linear assumption, the stiffness matrix of the particular element is appropriately modified and the problem is solved again using the new stiffness matrix. This method is known as *variable stiffness method*. In the other class, when the material behaviour of an element becomes nonlinear, instead of changing the original stiffness matrix an iterative approach using the same initial stiffness matrix is used. However, the

nonlinear effects are accounted for by means of the residual forces arising from equilibrium conditions. This method is known as *initial stiffness method*.

In both these methods either the analysis can be carried out by applying the total load or by using increments of total load. In general, the stresses and strains are dependent on the history of loading such as loading and unloading as well as cyclic loading. Therefore, it is preferable to use an incremental approach.

7.2 VARIABLE STIFFNESS METHOD

In the variable stiffness method, for an increment of load, the displacements, strains and stresses are calculated on the basis of linear elasticity

$$[K]\ \{u\} = \{P\} \tag{7.1}$$

where $[K]$ consists of element stiffness matrices given by

$$[k]^e = \int_V [B]^T\ [D]\ [B]\ dV \tag{7.2}$$

If it is found that in an element, the assumption of linear elasticity is not valid; then the stiffness matrix for that element is modified, such that

$$[k]^e_{n\ell} = \int_V [B]^T_{n\ell}\ [D]_{n\ell}\ [B]_{n\ell}\ dV \tag{7.3}$$

where $[B]_{n\ell}$ is the nonlinear relationship of strain-displacement

and $[D]_{n\ell}$ is the inelastic material matrix

Now, with this modified matrix, the stiffness matrix for the whole structure is assembled and the solution for displacements, stresses and strains are obtained

$$[K^*] \{u^*\} = \{P\} \tag{7.4}$$

It can be noted from the description of this method that its main disadvantage is that everytime an element behaviour is nonlinear, the stiffness matrix has to be reformulated and the entire system of equations is to be solved again. This is equivalent to solving different linear elastic problems. Computationally, this method will be, in most of the cases, uneconomical. In addition, this method fails for certain cases where, for example, the element behaviour is perfectly plastic.

7.3 INITIAL STIFFNESS METHOD

In the case of initial stiffness method, for an increment of load, even when the element behaviour is nonlinear, the same initial stiffness matrix as in Eq. (7.1) is used instead of the one given in Eq. (7.4). The nonlinear effects can be taken into account by means of nodal forces calculated on the basis of difference between Eq. (7.1) and (7.4). If the forces due to nonlinear effects are defined as

$$\{P_i\} = \int_V [B]^T_{n\ell} \{\sigma_i\} \, dV \tag{7.5}$$

where $\{\sigma_i\}$ is the initial stresses due to inelastic nature.

At this stage one can calculate the internal stresses due to the applied external load and the initial forces using the original stiffness matrix. In order to check for equilibrium conditions calculate the nodal forces due to internal stress distribution and compare with the external nodal forces. If the external nodal forces are $[P]$ and the internal forces are $\{P^*\}$, then the residual forces,

$$\{P_R\} = \{P\} - \{P^*\} \tag{7.6}$$

Therefore, in order to reduce the system of residual forces to zero, an iterative procedure is adopted, in which the initial forces $\{P_i\}$ are calculated for every iteration and applied to the problem along with the original stiffness matrix and the external loads. When the residual forces are negligible, convergences deemed to have obtained and the next load increment is applied.

As can be seen, the initial stiffness method does not require the modification of the initial stiffness matrix and the different loading conditions due to initial loads can be easily solved using an efficient solution system based on direct elimination. This will result in a more economical analysis.

In the following sections, only the material nonlinearity will be assumed for the sake of simplicity and different techniques based on the initial stiffness matrix will be described.

The linear elastic constitutive law can be written as

$$\{\sigma\} = \{\sigma_o\} + [D] (\{\varepsilon\} - \{\varepsilon_o\}) \qquad (7.7)$$

where $\{\sigma_o\}$ = initial stress

and $\{\varepsilon_o\}$ = initial strain

In order to conform with the appropriate nonlinear stress-strain relationship, if an adjustment in the linear law specified by Eq. (7.7) has to be made, it can be done by one of the following three ways:

a) the matrix [D] can be adjusted. In this case it will be equivalent to the variable stiffness method described in Section 7.2.

(or b) the initial strain can be adjusted, keeping the elasticity matrix [D] constant. This technique is known as *initial strain approach*.

(or c) the initial stress can be adjusted, keeping the elasticity matrix constant. This process is termed the *initial stress approach*.

7.3.1 INITIAL STRAIN APPROACH

This method is suitable if the initial strains such as due to creep can be determined directly from the stresses. The procedure is as follows:

a) for a typical increment of load or time, calculate the total incremental stresses and strains from the displacements using the original elasticity matrix.

b) compute the initial strain (creep, plasticity, etc)

c) the initial strain calculated in step (b) may not be correct because the stresses were calculated on the basis of linear law. Therefore, convert them into equivalent nodal forces.

$$\{d\sigma\} = [D]\ (\{d\varepsilon\} - \{d\varepsilon_0\}) \tag{7.8a}$$

$$\{dP_i\} = \int_V [B]^T\ [D]\ \{d\varepsilon_0\}\ dV \tag{7.8b}$$

d) for the next iteration, apply these initial loads and solve the problem again using the initial elasticity matrix.

e) check for convergence. If there is not appreciable difference between consecutive values of initial strains, convergence is obtained. Apply another increment of load or time. Otherwise repeat steps (b) to (d).

This approach is not applicable for the materials which are perfectly plastic or of low strain-hardening parameter. This can be seen from Fig 7.1.

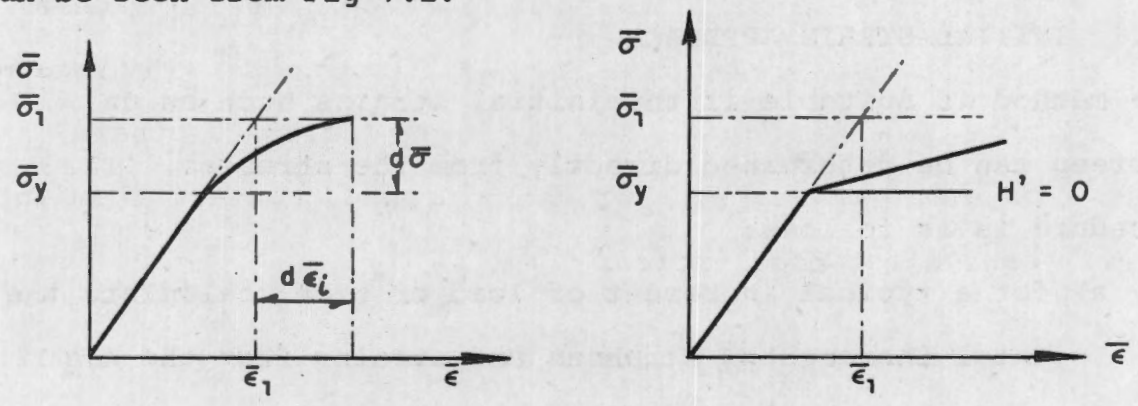


Fig 7.1

7.3.2 INITIAL STRESS APPROACH

In this process, similar steps as in the initial strain approach are carried out but the corrections are made to initial stresses. Thus,

$$\{d\sigma\} = \{d\sigma_o\} + [D]\ \{d\varepsilon\} \tag{7.9a}$$

$$\text{(or)}\quad \{d\sigma\} - \{d\sigma_o\} = [D]\ \{d\varepsilon\} \tag{7.9b}$$

The major steps are as follows:

a) for a typical increment of load, calculate the strains and stresses from the displacements using the original elasticity matrix.

b) for nonlinear behaviour, the stresses will be given by

$$\{d\sigma_1\} = [D^*]\ \{d\varepsilon_1\} \tag{7.10a}$$

However, using the original elasticity matrix

$$\{d\sigma_1\}' = [D]\ \{d\varepsilon_1\} \tag{7.10b}$$

Thus the difference between the two is

$$\{d\sigma_i\} = \{d\sigma_1\}' - \{d\sigma_1\} = ([D] - [D^*])\ \{d\varepsilon\} \tag{7.11}$$

This initial stress can be viewed upon as the excess stress which the material cannot withstand. Therefore, this excess stress has to be distributed to other adjacent elements. For this purpose, convert them into equivalent nodal forces

$$\{dP_i\} = \int_V [B]^T\ \{d\sigma_i\}\ dV \tag{7.12}$$

c) during the next iteration, apply these initial forces and use the original elasticity matrix.

d) repeat steps (b) and (c) unless convergence has been obtained. Convergence can be based upon the negligible initial forces. For this purpose either the absolute value of the largest initial force or the norm of the initial forces can be considered.

As can be seen from Fig 7.2, this method is applicable to all cases strain-hardening and perfectly plastic materials, strain-softening materials and cyclic loading situations.

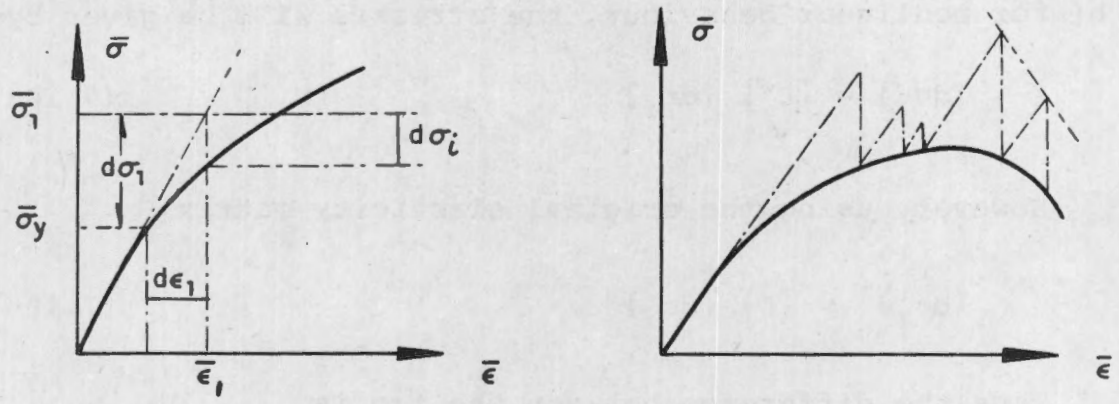


Fig 7.2

From Fig 7.2, it can be noted that the initial stress process converges very slowly in the case of strain-softening. Therefore, some accelerator techniques can be employed. In one such technique, the displacements calculated from the initial forces are used to calculate what is known as "alpha" constant.

$$\alpha_k = \frac{dw_k}{d\delta_k} + 1 \qquad (7.13)$$

$$\alpha_k = 1 \quad \text{if} \quad d\delta_k = 0 \tag{7.13a}$$

$$dw_k = [K]^{-1} \{dP_i\} \tag{7.14}$$

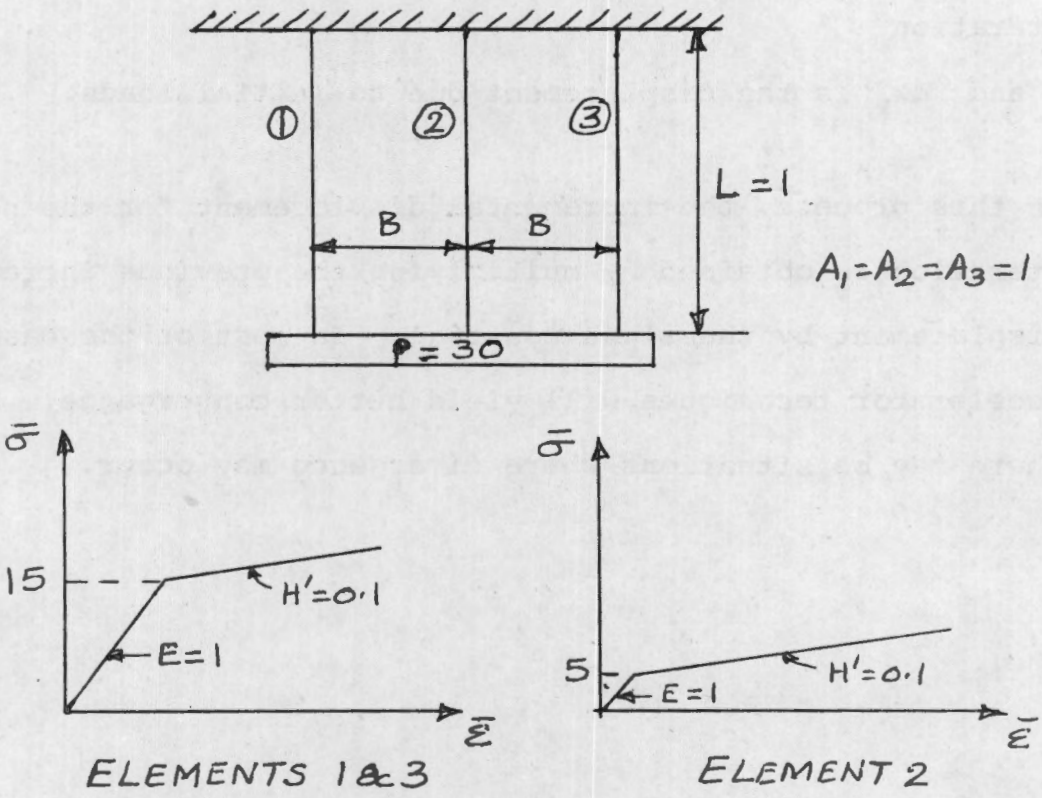
where $d\delta_k$ is the displacement due to $\{dP_i\}$ and $\{dP\}$ in k^{th} iteration

and dw_k is the displacement due to initial loads.

In this process, the incremental displacement for the next iteration is obtained by multiplying the previous incremental displacement by the alpha constant. In most of the cases, the accelerator techniques will yield better convergence. However, there may be situations where divergence may occur.

EXAMPLE 7.1

Analyse the three bar structure shown in the sketch using "variable stiffness method" and "initial stress method".



Variable stiffness method:

$$\delta_1 = \left(\frac{PL}{AE}\right)_1 = P_1 f_1 \quad \text{or } P_1 = k_1 \delta_1$$

$$\delta_1 = \delta_2 = \delta_3 \qquad \text{compatibility}$$

$$P_1 + P_2 + P_3 = P \qquad \text{equilibrium}$$

$$k_1 \delta_1 + k_2 \delta_2 + k_3 \delta_3 = P$$

$$\text{(or)} \quad \delta \left[k_1 + k_2 + k_3 \right] = P$$

Apply load increment of $\Delta P_1 = 15$

$$k_1 = k_2 = k_3 = \frac{AE}{L} = \frac{1 \times 1}{1} = 1$$

$$\delta = \frac{15}{3} = 5$$

$$dP_1 = dP_2 = dP_3 = 1 \times 5 = 5$$

Apply second increment of load, $\Delta P_2 = 15$

As before, $\delta = 5$ and $dP = 5$

Element 2 will yield. Change its stiffness matrix.

$$k_2 = \frac{1 \times 0.1}{1} = 0.1$$

$$\delta \left[1 + 0.1 + 1 \right] = 15$$

$$\delta = \frac{15}{2.1} = 7.14$$

$$dP_1 = dP_3 = 1 \times 7.14 = 7.14$$

$$dP_2 = 0.1 \times 7.14 = 0.714$$

Thus, $P_1 = P_3 = 5 + 7.14 = 12.14$

$$P_2 = 5 + 0.714 = 5.714$$

Initial stress method:

$$\{d\sigma_1\} = [D_{ep}]\ \{d\varepsilon\}$$

$$\{d\sigma_1\}' = [D]\ \{d\varepsilon\}$$

$$\{d\sigma_i\} = ([D] - [D_{ep}])\ \{d\varepsilon\}$$

For element 2 , $D = 1.0$ $D_{ep} = 0.1$

$D - D_{ep} = 0.9$

		P_1	P_2	P_3	
Inc. 1	$\Delta P = 15$	5	5	5	E = 1 for all
			(5.0)		
Inc. 2	$\Delta P = 15$	5	0.5	5	5 × 0.1 = 0.5
			(1.5)		distribute 4.5
Iter.	1	1.5	0.15	1.5	distribute 1.35
			(0.45)		
	2	0.45	0.045	0.45	distribute 0.405
			(0.14)		
	3	0.135	0.014	0.135	distribute 0.121
			(0.04)		
	4	0.040	0.004	0.040	distribute 0.036
			(0.012)		
	5	0.012	0.012	0.012	
		12.137	5.725	12.137	

8. NONLINEAR ANALYSIS - MATERIAL NONLINEARITY

8.1 INTRODUCTION

In this chapter, the discussion will be confined to problems with nonlinear material behaviour. Among the various nonlinear constitutive relationships, only those connected with plasticity, creep and tension cut-off will be described in detail. The ways by which these three material nonlinearities can be handled in FEM will be explained with reference to initial stiffness method. Finally, the various nonlinear constitutive laws available will be discussed.

8.2 PLASTICITY

The plastic behaviour is independent of time but is dependent on the stress level. In order to analyse an elasto-plastic problem, it is necessary to define

1) yield criterion which specifies the onset of plastic flow
2) elastic stress-strain relationship and
3) plastic stress-strain relationship for which an appropriate flow rule has to be used to determine the plastic strain

8.2.1 YIELD CRITERION

The yield criterion is a condition specifying a particular combination of stress components, which initiates the plastic flow. Thus it is a function of not only the stresses but also the hardening parameter which defines the modified yield surface during the

plastic flow. Therefore, the general form of an yield criterion can be expressed as

$$F(\{\sigma\}, \kappa) = 0 \tag{8.1}$$

$$\text{(or)} \quad F(J_1, J_2, J_3) = \kappa \tag{8.2}$$

where

$$J_1 = \text{First stress invariant} = \sigma_x + \sigma_y + \sigma_z$$

$$J_2 = \text{Second stress invariant} = \sigma_x \sigma_y + \sigma_y \sigma_z + \sigma_z \sigma_x - \tau_{xy}^2 - \tau_{yz}^2 - \tau_{zx}^2 \tag{8.3}$$

$$J_3 = \text{Third stress invariant} = \sigma_x \sigma_y \sigma_z - \sigma_x \tau_{yz}^2 - \sigma_y \tau_{zx}^2 - \sigma_z \tau_{xy}^2 + 2\tau_{xy} \tau_{yz} \tau_{zx}$$

and κ = hardening parameter which is a function of strain.

When yielding is independent of the hydrostatic stress, Eq.(8.2) can be written as

$$F(J_2', J_3') = \kappa \tag{8.4}$$

where

J_2' is 2nd invariant of deviatoric stresses

$$= \frac{J_1^2}{3} - J_2$$

and J_3' is 3rd invariant of deviatoric stresses

$$= \frac{2J_1^3}{27} - \frac{J_1 J_2}{3} + J_3$$

8.2.2 PLASTIC POTENTIAL

In the incremental or flow theory of plasticity, it is postulated that the plastic strain increment $\{d\varepsilon^p\}$ can be obtained from the differential of the plastic potential which is defined by

$$g\ (\{\sigma\},\ \kappa) = 0 \tag{8.5}$$

$$\{d\varepsilon^p\} = d\lambda\ \{\frac{\partial g}{\partial \sigma}\} \tag{8.6}$$

where $d\lambda$ is known as the proportionality constant

The principle on which Eq. (8.6) is based is called the normality principle which states that if the yield surface is assumed to be smooth and convex, then the plastic strain vector at any point on the yield surface is outward normal to the tangent at that point.

If the plastic potential defined by Eq. (8.5) and the yield criterion given by Eq. (8.1) are the same, the flow rule is termed the associated flow rule whereas it is known as the nonassociated flow rule, if they are different.

On the basis of associated flow rule, the plastic strain increment can be obtained as

$$\{d\varepsilon^p\} = d\lambda\ \{\frac{\partial F}{\partial \sigma}\} \tag{8.7}$$

$$\text{If we let } \{b\} = \{\frac{\partial F}{\partial \sigma}\} \tag{8.8a}$$

$$\text{then } \{d\varepsilon^p\} = d\lambda\ \{b\} \tag{8.8b}$$

8.2.3 SPECIAL FORMS OF YIELD CRITERIA

a) Tresca's criterion:

Tresca's criterion is based on the maximum shear stress and can be written as

$$\sigma_1 - \sigma_3 = \kappa \; ; \quad \sigma_1 > \sigma_2 > \sigma_3 \tag{8.9}$$

where κ is the yield stress in shear

b) von Mises criterion:

von Mises criterion is based on the maximum distortion energy and can be written as

$$F = \sqrt{3J_2'} = \sqrt{3}\ \bar{\sigma} = \sigma_y \tag{8.10}$$

where

$$\bar{\sigma} = \text{effective stress} = \sqrt{J_2'}$$

$$= \sqrt{\frac{1}{6}} \left[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6\,(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2) \right]^{1/2} \tag{8.11}$$

and σ_y = is the uniaxial yield stress

For the particular case of uniaxial stress field,

$$\bar{\sigma} = \sqrt{\frac{1}{6}\, 2\sigma_x^2} = \sqrt{\frac{1}{3}}\ \sigma_x = \sigma_y \tag{8.12}$$

both Tresca's and von Mises, yield criteria, have been extensively used for metals. However, for materials such as soils and rocks, it is necessary to express an yield criterion which includes parameters defining the cohesion and friction properties. Thus, Mohr-Coulomb and Drucker-Prager criteria are considered to be more suitable for these materials.

c) Mohr-Coulomb criterion:

This failure criterion is generally expressed as

$$\tau \leq c + \sigma_n \tan \phi \tag{8.13}$$

where

τ is the shear stress

σ_n is the normal stress

c is the cohesion

and ϕ is the angle of internal friction

d) Drucker-Prager criterion:

Drucker-Prager criterion is the modified form of von Mises criterion on the basis of Mohr-Coulomb failure criterion. It can be written as

$$F = \alpha J_1 + \sqrt{J_2'} = \kappa \tag{8.14}$$

where α and κ are material constants depending on cohesion and friction

$$\alpha = \frac{2 \sin \phi}{\sqrt{3}\ (3 - \sin \phi)} \tag{8.15}$$

and $$\kappa = \frac{6c \cos \phi}{\sqrt{3}\,(3 - \sin \phi)}$$

e) Hill's criterion for anisotropic materials:

For orthotropic materials, Hill extended the von Mises criterion as follows

$$F = \bar{\sigma} = \{ \frac{1}{2} \left[\alpha_{12} (\sigma_x - \sigma_y)^2 + \alpha_{23}(\sigma_y - \sigma_z) + \alpha_{31}(\sigma_z - \sigma_x)^2 \right] + 3\left[\alpha_{44} \tau_{xy}^2 + \alpha_{55} \tau_{yz}^2 + \alpha_{66} \tau_{zx}^2\right] \}^{1/2} \quad (8.16)$$

in which α's are anisotropic parameters

f) Extension of Hill's criterion to geomechanics:

For soils and rocks, Pariseu extended Hill's criterion as given below

$$F = \left[\alpha_{12} (\sigma_x - \sigma_y)^2 + \alpha_{23}(\sigma_y - \sigma_z)^2 + \alpha_{31}(\sigma_z - \sigma_x)^2 + \alpha_{44} \tau_{xy}^2 + \alpha_{55} \tau_{yz}^2 + \alpha_{66} \tau_{zx}^2 \right]^{1/2} + (\alpha_{11} \sigma_x + \alpha_{22} \sigma_y + \alpha_{33} \sigma_z) = \kappa \quad (8.17)$$

8.2.4 ELASTIC-PLASTIC STRESS-STRAIN RELATIONSHIP

In the incremental theory of plasticity, the total strain for a typical increment of load, is written as the combination of elastic and plastic strains

$$\{d\varepsilon\} \quad \{d\varepsilon^e\} + \{d\varepsilon^p\} \quad (8.18)$$

where $\{d\varepsilon^e\} = [D^e]^{-1} \{d\sigma\}$ (8.19a)

and $\{d\varepsilon^p\} = d\lambda \{b\}$ (8.19b)

Thus, the total strain for an increment is

$$\{d\varepsilon\} = [D^e]^{-1} \{d\sigma\} + d\lambda \{b\} \tag{8.20}$$

The constant $d\lambda$ can be obtained by differentiating the yield criterion, F

$$\frac{\partial F}{\partial \kappa} d\kappa = \{\frac{\partial F}{\partial \sigma}\}^T \{d\sigma\} \tag{8.21}$$

Introducing a parameter A such that

$$A = \frac{\partial F}{\partial \kappa} d\kappa \frac{1}{d\lambda} \tag{8.22}$$

we can write Eq. (8.21) as

$$\{b\}^T \{d\sigma\} - A d\lambda = 0 \tag{8.23a}$$

Thus, the total stress-strain relationship can be written as

$$\begin{bmatrix} d\varepsilon \\ 0 \end{bmatrix} = \begin{bmatrix} [D^e]^{-1} & \{b\} \\ \{b\}^T & -A \end{bmatrix} \begin{bmatrix} d\sigma \\ \lambda \end{bmatrix} \tag{8.23b}$$

Premultiplying both sides of Eq. (8.20) by $\{b\}^T [D^e]$,

$$\{b\} [D^e] \{d\varepsilon\} = \{b\}^T [D^e] ([D^e]^{-1} \{d\sigma\} + d\lambda \{b\}) \tag{8.24}$$

Using Eq.(8.23a) in Eq. (8.24)

$$\{b\}^T [D^e] \{d\varepsilon\} = A d\lambda + \{b\}^T [D^e] \{b\} d\lambda \tag{8.25}$$

$$\text{or} \quad d\lambda = \left[A + \{b\}^T [D^e] \{b\} \right]^{-1} \{b\}^T [D^e] \{d\varepsilon\} \qquad (8.26)$$

Substituting this value of $d\lambda$ in Eq. (8.20),

$$[D^e]^{-1} \{d\sigma\} = \{d\varepsilon\} - \left[A + \{b\}^T [D^e] \{b\} \right]^{-1} \{b\}\{b\}^T [D^e] \{d\varepsilon\} \qquad (8.27)$$

$$\text{or} \quad \{d\sigma\} = ([D^e] - [D^p]) \{d\varepsilon\}$$

$$= [D^{ep}] \{d\varepsilon\} \qquad (8.28)$$

$$\text{where } [D^p] = [D^e] \{b\} \{b\}^T [D^e] \left[A + \{b\}^T [D^e] \{b\} \right]^{-1} \qquad (8.29)$$

The elasto-plastic matrix $[D^{ep}]$ is symmetric for associated flow rules and is positive definite when A is positive.

It can be seen from the definition of "A" in Eq. (8.22) that its significance depends on the nature of the hardening rule. For ideally plastic materials, it is zero. For isotropic hardening materials, its value can be determined as follows:

the amount of plastic work done during plastic deformation is

$$du = d\kappa = \{\sigma\}^T \{d\varepsilon^p\} = \{\sigma\}^T \{b\} \, d\lambda \qquad (8.30)$$

substituting Eq. (8.30) into Eq. (8.22),

$$A = \frac{\partial F}{\partial \kappa} \{\sigma\}^T \{b\} \qquad (8.31)$$

Now the value of A can be obtained using the relationship between the yield criterion F and the hardening parameter κ.

Using the effective stress and the effective plastic strain, the plastic work done can be written as

$$dU = d\kappa = \bar{\sigma}\, d\bar{\varepsilon}^{p} \tag{8.32}$$

From Eq. (8.30) and (8.32),

$$d\lambda = \frac{\bar{\sigma}\, d\bar{\varepsilon}^{p}}{\{\sigma\}^{T}\{b\}} \tag{8.33}$$

For the unixial case, the effective plastic strain can be written as

$$d\bar{\varepsilon}^{p} = \frac{d\bar{\sigma}}{H'} \tag{8.34}$$

where H' is the slope of the effective stress-effective plastic strain curve.

Also, $$\frac{dF}{d\kappa} = \frac{d\bar{\sigma}}{d\kappa} \tag{8.35}$$

Thus, from Eq. (8.22)

$$A = \frac{\partial F}{\partial \kappa}\, d\kappa\, \frac{1}{d\lambda}$$

$$= \frac{d\bar{\sigma}}{d\kappa}\, d\kappa\, \frac{\{\sigma\}^{T}\{b\}}{\bar{\sigma}\, d\bar{\varepsilon}^{p}}$$

$$= \frac{d\bar{\sigma}}{d\kappa}\, d\kappa\, \frac{\{\sigma\}^{T}\{b\}\, H'}{\bar{\sigma}\, d\bar{\sigma}}$$

$$A = \{\sigma\}^{T}\{b\}\, \frac{H'}{\bar{\sigma}} \tag{8.36}$$

In the case of von Mises yield criterion it can be shown that

$$\{\sigma\}^T \{b\} = \bar{\sigma} \tag{8.37}$$

Therefore the parameter "A" reduces to

$$A = H' \tag{8.38}$$

and $$d\lambda = d\bar{\varepsilon}^p \tag{8.39}$$

As can be seen from Eq. (8.28) it is necessary to use the elasto-plastic matrix $[D^{ep}]$ if an element becomes plastic during a particular increment of load. This will involve calculating the new stiffness matrix for the element as well as renewed assembly and solution. However, if the initial stress method, as described in the previous chapter, is used it is only necessary to calculate the initial stress due to plasticity. This initial stress can be obtained from Eq.(8.28)

$$\{d\sigma_i\} = [D^p] \{d\varepsilon\} \tag{8.40}$$

In calculating $[D^p]$, it is necessary only to form $\{b\}$ for every iteration since $[D^e]$ is the original elasticity matrix. It is also worth noting that Eq. (8.29) is valid both for ideal plasticity (A = 0) and for work-hardening materials.

8.3 TENSION CUT-OFF

Materials such as rock, concrete and soil have only a limited tensile strength or none at all. Therefore in dealing with such materials, when tensile stresses are developed in any part of the structure or continuum made up of these materials, such excess tensile stresses should be redistributed to other parts. In fact, such stress transfer occurs in a physical sense. In the finite element analysis, this process can be carried out as follows:

1. for a typical increment of load, calculate the displacements and then the stresses. Determine the principal stresses σ_1 and σ_2 (two-dimensional cases).
2. check to see whether these principal stresses exceed the limited tensile strength of the material, $\sigma_{\ell t}$. The value of $\sigma_{\ell t}$ is as prescribed initially for the element before cracking and once the element has cracked the value of $\sigma_{\ell t}$ will be zero.
3. determine the excess tensile stress and the stresses to be distributed,

$$\begin{aligned} \sigma_1' &= \sigma_1 - \sigma_{\ell t} \\ \sigma_2' &= \sigma_2 - \sigma_{\ell t} \end{aligned} \tag{8.41}$$

$$\{\sigma_t\} = \begin{bmatrix} \sigma_x^t \\ \sigma_y^t \\ \tau_{xy}^t \end{bmatrix} = \begin{bmatrix} \text{Cos}^2\,\theta & \text{Sin}^2\,\theta \\ \text{Sin}^2\,\theta & \text{Cos}^2\,\theta \\ \frac{1}{2}\,\text{Sin}\,2\theta & -\frac{1}{2}\,\text{Sin}\,2\theta \end{bmatrix} \begin{bmatrix} \sigma_1' \\ \sigma_2' \end{bmatrix} \tag{8.42}$$

4. from the stresses to be distributed, calculate the equivalent nodal forces

$$\{F_i\}^e = \int_V [B]^T \{\sigma^t\}\, dV \tag{8.43}$$

5. apply these nodal forces for the next iteration and repeat steps 1 to 4 until convergence is obtained. The original elasticity matrix is used for every increment and iteration.

6. if convergence is not obtained within a maximum number of iterations prescribed, then the structure is considered to be at the verge of failure.

8.4 CREEP

The creep strain is caused due to the time-dependent nonlinear effects whereas the plastic strain is a time-independent phenomena. In order to assess the creep strain, the initial strain approach can be employed, as corrections in the initial strain of Eq. (7.7) can be used just like the corrections in the initial stress were used for elasto-plastic analysis. Again due to the time dependent nature, the increments can be in the time domain rather than the load increments.

Within a typical time increment, the total incremental strain in an element can be divided into elastic and creep strains as

$$\{d\varepsilon\} = \{d\varepsilon^e\} + \{d\varepsilon^c\} \tag{8.44}$$

where $$\{d\varepsilon^e\} = [D^e]^{-1} \{d\sigma\} \tag{8.45}$$

the incremental creep strain is generally a function of the effective stress $\bar{\sigma}$, the total effective creep strain $\bar{\varepsilon}^c$, and the time, t. Thus, it can be expressed in a functional form as

$$\{d\varepsilon^c\} = F\ (\bar{\sigma},\ \bar{\varepsilon}^c,\ t) \tag{8.46}$$

The effective stress is same as defined for plasticity problems. The total effective creep strain depends on the creep law assumed for a particular material.

Having determined the incremental creep strains, the incremental stresses can be obtained as

$$\{d\sigma\} = [D^e]\ (\{d\varepsilon\} - \{d\varepsilon_i\}) \tag{8.47}$$

8.4.1 CREEP LAWS

a) Exponential law.- $\bar{\varepsilon}^c = A\ (1 - e^{Bt})$ (8.48)

b) Hyperbolic law.- $\bar{\varepsilon}^c = \frac{ct}{D+t}$ (8.49)

where A, B, C and D are constants depending on the stress level and exposure conditions.

c) $\bar{\varepsilon}^c = A\ e^{\alpha m}\ t^n$ for soils (8.50)

where $A = \frac{x}{n}$ (8.50a)

$$m = \frac{\sigma_1 - \sigma_3}{2c} \tag{8.50b}$$

c is cohesion

and α, x, n are constants

8.4.2 SELECTION OF TIME INCREMENT

Since the stresses are assumed to remain constant in any time interval (in the initial strain process), the selection of a suitable time increment is critical in the analysis. If large increments are used, the solution is likely to diverge. On the other hand, if small time increments are used, it will involve more computer time. Thus, it is necessary to adopt an optimal time increment. One criterion for choosing such time increment would be to keep the creep strains smaller than the elastic strains within an increment. If incremental creep strain exceed the elastic strain, divergence is likely to occur.

8.5 NONLINEAR ELASTICITY

In soil mechanics problems, various investigators have been adopting piecewise linearization to represent the nonlinear behaviour of soils. For this purpose, the tangent stiffness method (variable stiffness) is quite suitable. The tangent modulus is chosen on the basis of the stress level in an element. The various constitutive relationships which have been used can be summarized as follows:

a) <u>tangent modulus</u>

$$E_t = \left[1 - \frac{R_f\ (1 - \text{Sin}\ \phi)(\sigma_1 - \sigma_3)}{2c\ \text{Cos}\ \phi + 2\sigma_3\ \text{Sin}\ \phi} \right] \kappa\ P_a\ \left(\frac{\sigma_3}{P_a}\right)^n \qquad (8.51)$$

tangent Poisson's ratio

$$\nu_t = \frac{G - F \log (\sigma_3/P_a)}{(1 - A)^2} \tag{8.52}$$

where R_f is failure ratio which is less than unity

c and ϕ are Mohr-Coulomb strength parameters

κ and n are experimentally determined parameters

P_a is atmospheric pressure

$$A = \frac{(\sigma_1 - \sigma_3)\, d}{\kappa\, P_a (\sigma_3/P_a)^n \left[1 - \frac{R_f\,(\sigma_1 - \sigma_3)(1 - \sin\phi)}{2c\cos\phi + 2\sigma_3 \sin\phi}\right]} \tag{8.53}$$

G and F are parameters determined experimentally

b) $$E_t = E_o\,(1 + \beta J_1) \qquad \text{for sand} \tag{8.54}$$

$$E_t = E_o\,(1 - \alpha \sqrt{J_2}) \qquad \text{for clay} \tag{8.55}$$

where α and β are nonlinear coefficients

J_1 and J_2 are first and second stress invariants

and E_o is the initial modulus

c) <u>cubic spline functions</u>

$$S(\sigma) = [N]^T \{q\} \tag{8.56}$$

$$[q]^T = \left[\, \varepsilon_{i-1},\ \varepsilon_i,\ \phi_{i-1},\ \phi \,\right]$$

$[\phi]$ = Second derivative of S (σ)

$$[N]^T = \frac{1}{6\lambda_i}\left[\, 6\,(\sigma_i - \sigma) \quad 6\,(\sigma - \sigma_{i-1}) \quad (\sigma_i - \sigma)^3 \right.$$

$$\left. - \lambda_i^2\,(\sigma_i - \sigma)(\sigma - \sigma_{i-1})^3 - \lambda_i^2\,(\sigma - \sigma_{i-1}) \,\right]$$

$$\lambda_i = \sigma_i - \sigma_{i-1}$$

$$E_t = \frac{1}{S'(\sigma)} \tag{8.57}$$

and $S'(\sigma) = [N']^T \{q\}$ (8.58)

F/DEPFI/No.5/1979/EJ.3
702456