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

SCHOOL OF MECHANICAL ENGINEERING

DEPARTMENT OF AERONAUTICAL ENGINEERING

UNIT – I– FINITE ELEMENT METHOD FOR AIRCRAFT STRUCTURES – SAEA1504

UNIT1 - INTRODUCTION

1.1.1 Introduction

The Finite Element Method (FEM) is a numerical technique to find approximate solutions of partial differential equations. It was originated from the need of solving complex elasticity and structural analysis problems in Civil, Mechanical and Aerospace engineering. In a structural simulation, FEM helps in producing stiffness and strength visualizations. It also helps to minimize materialweight and its cost of the structures. FEM allows for detailed visualization and indicates the distribution of stresses and strains inside the body of a structure. Many of FE software are powerful yet complex tool meant for professional engineers with the training and education necessary to properly interpret the results.

Several modern FEM packages include specific components such as fluid, thermal, electromagnetic and structural working environments. FEM allows entire designs to be constructed, refined and optimized before the design is manufactured. This powerful design tool has significantly improved both the standard of engineering designs and the methodology of the design process in many industrial applications. The use of FEM has significantly decreased the time to take products from concept to the production line. One must take the advantage of the advent of faster generation of personal computers for the analysis and design of engineering product with precision level of accuracy.

1.1.2 Background of Finite Element Analysis

The finite element analysis can be traced back to the work by Alexander Hrennikoff (1941)and Richard Courant(1942). Hrenikoff introduced the framework method, in which a plane elastic medium was represented as collections of bars and beams.These pioneers share one essential characteristic: mesh discretization of a continuous domain into a set of discrete sub-domains, usually called elements.

- In 1950s, solution of large number of simultaneous equations became possible because of the digitalcomputer.
- In 1960, Ray W. Clough first published a paper using term “Finite Element Method”.
- In 1965, First conference on “finite elements” was held.
- In 1967, the first book on the “Finite Element Method” was published by Zienkiewicz and Chung.
- In the late 1960s and early 1970s, the FEM was applied to a wide variety of engineering problems.

- In the 1970s, most commercial FEM software packages (ABAQUS, NASTRAN, ANSYS, etc.) originated.Interactive FE programs on supercomputer lead to rapid growth of CAD systems.
- In the 1980s, algorithm on electromagnetic applications, fluid flow and thermal analysis were developed with the use of FE program.
- Engineers can evaluate ways to control the vibrations and extend the use of flexible, deployablestructures in space using FE and other methods in the 1990s. Trends to solve fully coupled solution of fluid flows with structural interactions, bio-mechanics related problems with a higher level of accuracy were observed in this decade.

With the development of finite element method, together with tremendous increases in computing power and convenience, today it is possible to understand structural behavior with levels of accuracy. This was in fact the beyond of imagination before the computer age.

1.1.3 Numerical Methods

The formulation for structural analysis is generally based on the three fundamental relations: equilibrium, constitutive and compatibility. There are two major approaches to the analysis: Analytical and Numerical. Analytical approach which leads to closed-form solutions is effective in case of simple geometry, boundary conditions, loadings and material properties. However, in reality, such simple cases may not arise. As a result, various numerical methods are evolved for solving such problems which are complex in nature. For numerical approach, the solutions will be approximate when any of these relations are only approximately satisfied. The numerical method depends heavily on the processing power of computers and is more applicable to structures of arbitrary size and complexity. It is common practice to use approximate solutions of differential equations as the basis for structural analysis. This is usually done using numerical approximation techniques. Few numerical methods which are commonly used to solve solid and fluid mechanics problems are given below.

- Finite Difference Method
- Finite Volume Method
- Finite Element Method
- Boundary Element Method
- Meshless Method

The application of finite difference method for engineering problems involves replacing the governing differential equations and the boundary condition by suitable algebraic equations. For example in the analysis of beam bending problem the differential equation is reduced to be solution of algebraic equations written at every nodal point within the beam member. For example, the beam equation can be expressed as:

$$\frac{d^4 w}{dx^4} = \frac{q}{EI} \qquad (1.1.1)$$

To explain the concept of finite difference method let us consider a displacement function variable namely $w = f(x)$

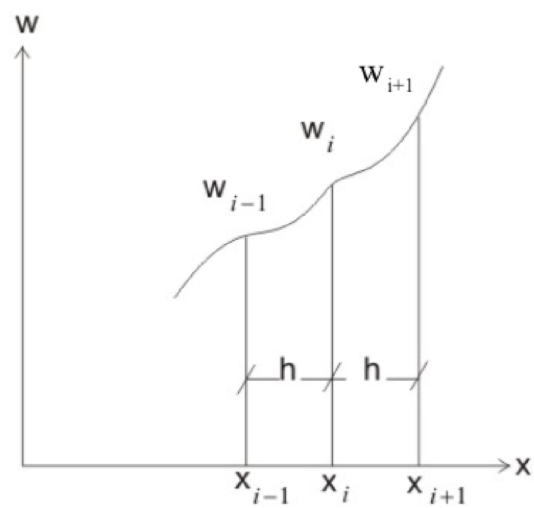


Fig. 1.1.1 Displacement Function

Now, $\Delta w = f(x + \Delta x) - f(x)$

So, $$\frac{dw}{dx} = \operatorname*{Lt}_{\Delta x \to 0} \frac{\Delta w}{\Delta x} = \operatorname*{Lt}_{\Delta x \to 0} \frac{f(x+\Delta x) - f(x)}{\Delta x} = \frac{1}{h}\left(w_{i+1} - w_i\right) \qquad (1.1.2)$$

Thus,

$$\frac{d^2 w}{dx^2} = \frac{d}{dx}\left[\frac{1}{h}\left(w_{i+1} - w_i\right)\right] = \frac{1}{h^2}\left(w_{i+2} - w_{i+1} - w_{i+1} + w_i\right) = \frac{1}{h^2}\left(w_{i+2} - 2w_{i+1} + w_i\right) \qquad (1.1.3)$$

$$\frac{d^3 w}{dx^3} = \frac{1}{h^3}\left(w_{i+3} - w_{i+2} - 2w_{i+2} + 2w_{i+1} + w_{i+1} - w_i\right)$$
$$= \frac{1}{h^3}\left(w_{i+3} - 3w_{i+2} + 3w_{i+1} - w_i\right) \qquad (1.1.4)$$

$$\frac{d^4w}{dx^4} = \frac{1}{h^4}\left(w_{i+4} - w_{i+3} - 3w_{i+3} + 3w_{i+2} + 3w_{i+2} - 3w_{i+1} - w_{i+1} + w_i\right)$$
$$= \frac{1}{h^4}\left(w_{i+4} - 4w_{i+3} + 6w_{i+2} - 4w_{i+1} + w_i\right) \qquad (1.1.5)$$
$$= \frac{1}{h^4}\left(w_{i+2} - 4w_{i+1} + 6w_i - 4w_{i-1} + w_{i-2}\right)$$

Thus, eq. (1.1.1) can be expressed with the help of eq. (1.1.5) and can be written in finite difference form as:

$$(w_{i-2} - 4w_{i-1} + 6w_i - 4w_{i+1} + w_{i+2}) = \frac{q}{EI}h^4 \qquad (1.1.6)$$

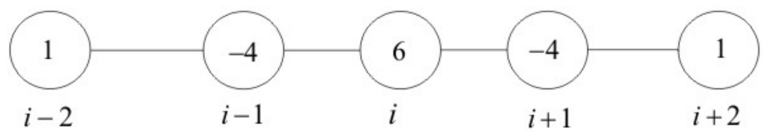


Fig. 1.1.2 Finite difference equation at node i

Thus, the displacement at node *i* of the beam member corresponds to uniformly distributed load can be obtained from eq. (1.1.6) with the help of boundary conditions. It may be interesting to note that, the concept of node is used in the finite difference method. Basically, this method has an array of grid points and is a point wise approximation, whereas, finite element method has an array of small interconnecting sub-regions and is a piece wise approximation.

Each method has noteworthy advantages as well as limitations. However it is possible to solve various problems by finite element method, even with highly complex geometry and loading conditions, with the restriction that there is always some numerical errors. Therefore, effective and reliable use of this method requires a solid understanding of its limitations.

1.1.4 Concepts of Elements and Nodes

Any continuum/domain can be divided into a number of pieces with very small dimensions. These small pieces of finite dimension are called ‘Finite Elements’ (Fig. 1.1.3). A field quantity in each element is allowed to have a simple spatial variation which can be described by polynomial terms. Thus the original domain is considered as an assemblage of number of such small elements. These elements are connected through number of joints which are called ‘Nodes’. While discretizing the structural system, it is assumed that the elements are attached to the adjacent elements only at the nodal points. Each element contains the material and geometrical properties. The material properties inside an element are assumed to be constant. The elements may be 1D elements, 2D elements or 3D elements. The physical object can be modeled by choosing appropriate element such as frame element, plate element, shell element, solid element, etc. All elements are then assembled to obtain the solution of the entire domain/structure under certain loading conditions. Nodes are assigned at a certain density throughout the continuum depending on the anticipated stress levels of a particular domain. Regions which will receive large amounts of stress variation usually have a higher node density than those which experience little or no stress.

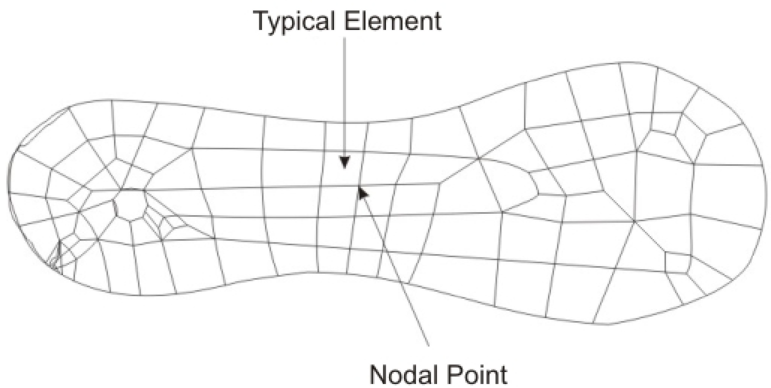


Fig. 1.1.3 Finite element discretization of a domain

1.1.5 Degrees of Freedom

A structure can have infinite number of displacements. Approximation with a reasonable level of accuracy can be achieved by assuming a limited number of displacements. This finite number of displacements is the number of degrees of freedom of the structure. For example, the truss member will undergo only axial deformation. Therefore, the degrees of freedom of a truss member with respect to its own coordinate system will be one at each node. If a two dimension structure is modeled by truss elements, then the deformation with respect to structural coordinate system will be two and therefore degrees of freedom will also become two. The degrees of freedom for various types of element are shown in Fig. 1.1.4 for easy understanding. Here (u, v, w) and $(\theta_x, \theta_y, \theta_z)$ represent displacement and rotation respectively.

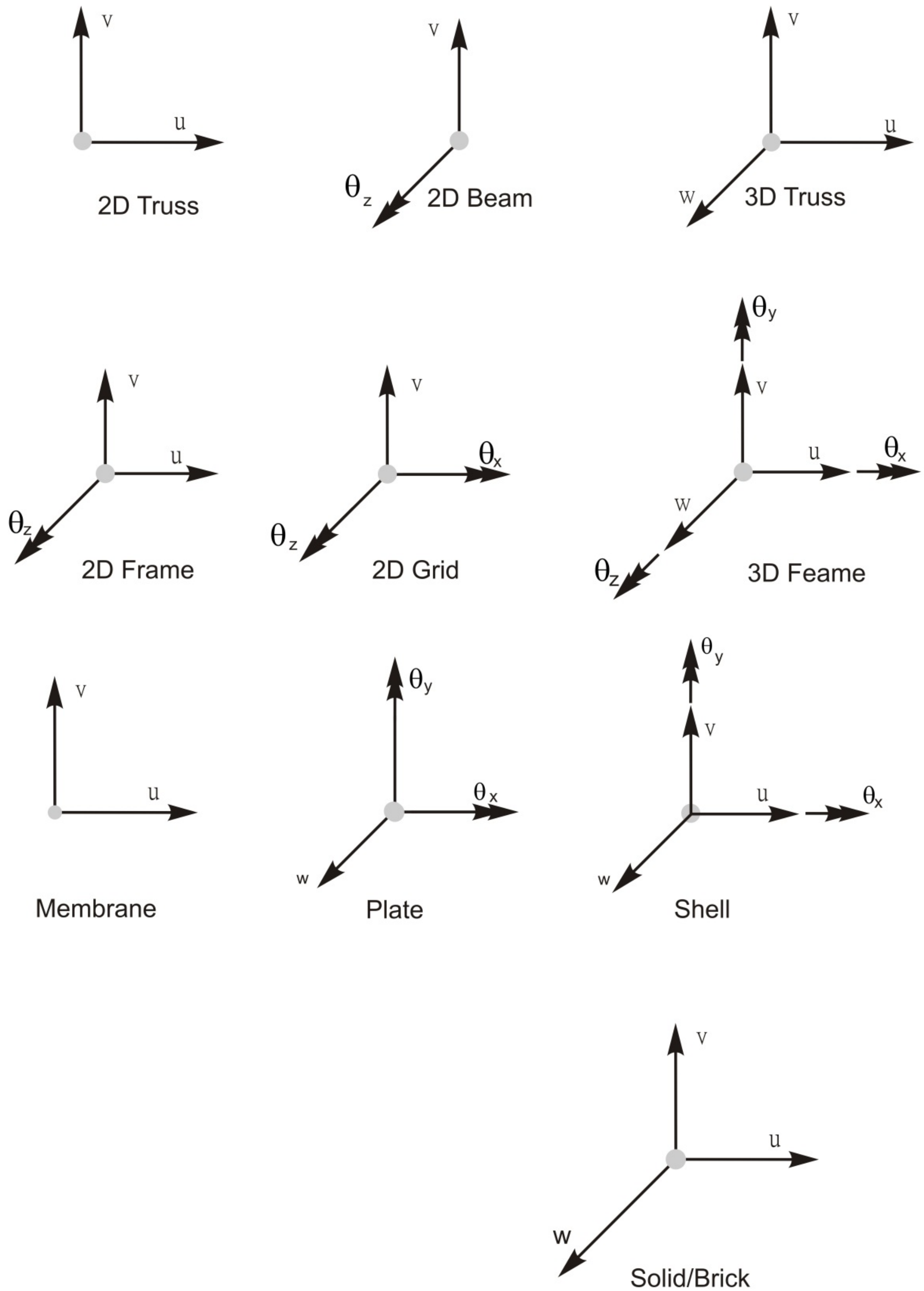


Fig. 1.1.4 Degrees of Freedom for Various Elements

Basic Concepts of Finite Element Analysis

1.2.1 Idealization of a Continuum

A continuum may be discretized in different ways depending upon the geometrical configuration of the domain. Fig. 1.2.1 shows the various ways of idealizing a continuum based on the geometry.

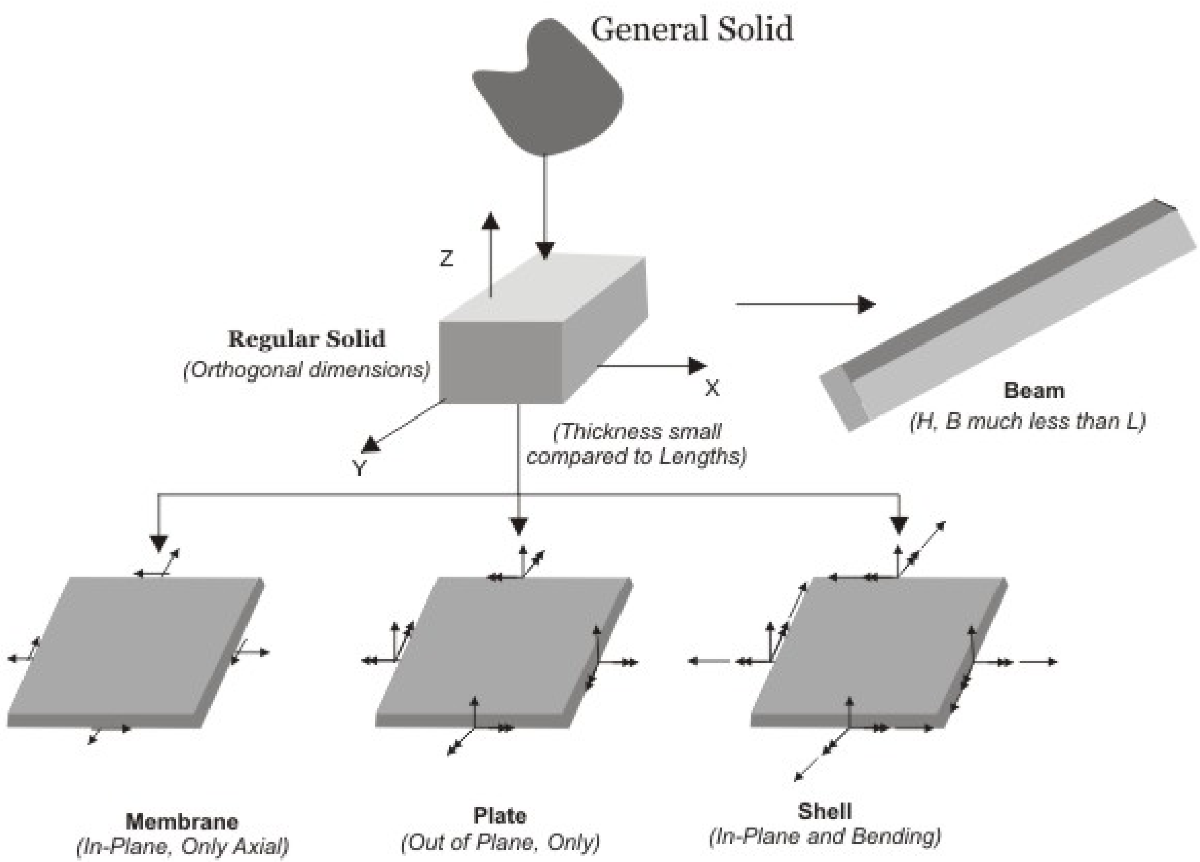


Fig. 1.2.1 Various ways of Idealization of a Continuum

1.2.2 Discretization of Technique

The need of finite element analysis arises when the structural system in terms of its either geometry, material properties, boundary conditions or loadings is complex in nature. For such case, the whole structure needs to be subdivided into smaller elements. The whole structure is then analyzed by the assemblage of all elements representing the complete structure including its all properties.

The subdivision process is an important task in finite element analysis and requires some skill and knowledge. In this procedure, first, the number, shape, size and configuration of elements have to be decided in such a manner that the real structure is simulated as closely as possible. The discretization is to be in such that the results converge to the true solution. However, too fine mesh will lead to extra computational effort. Fig. 1.2.2 shows a finite element mesh of a continuum using triangular and quadrilateral elements. The assemblage of triangular elements in this case shows better representation of the continuum. The discretization process also shows that the more accurate representation is possible if the body is further subdivided into some finer mesh.

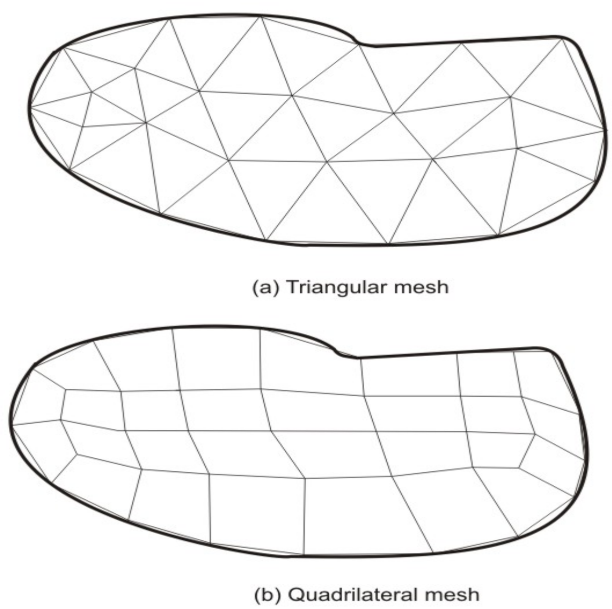


Fig. 1.2.2 Discretization of a continuum

1.2.3 Concepts of Finite Element Analysis

FEA consists of a computer model of a continuum that is stressed and analyzed for specific results. A continuum has infinite particles with continuous variation of material properties. Therefore, it needs to simplify to a finite size and is made up of an assemblage of substructures, components and members. Discretization process is necessary to convert whole structure to an assemblage of members/elements for determining its responses. Fig. 1.2.3 shows the process of idealization of actual structure to a finite element form to obtain the response results. The assumptions are required to be made by the experienced engineer with finite element background for getting appropriate response results. On the basis of assumptions, the appropriate constitutive model can be constructed.

For the linear-elastic-static analysis of structures, the final form of equation will be made in the form of *F=Kd*where *F*, *K* and *d* are the nodal loads, global stiffness and nodal displacements respectively.

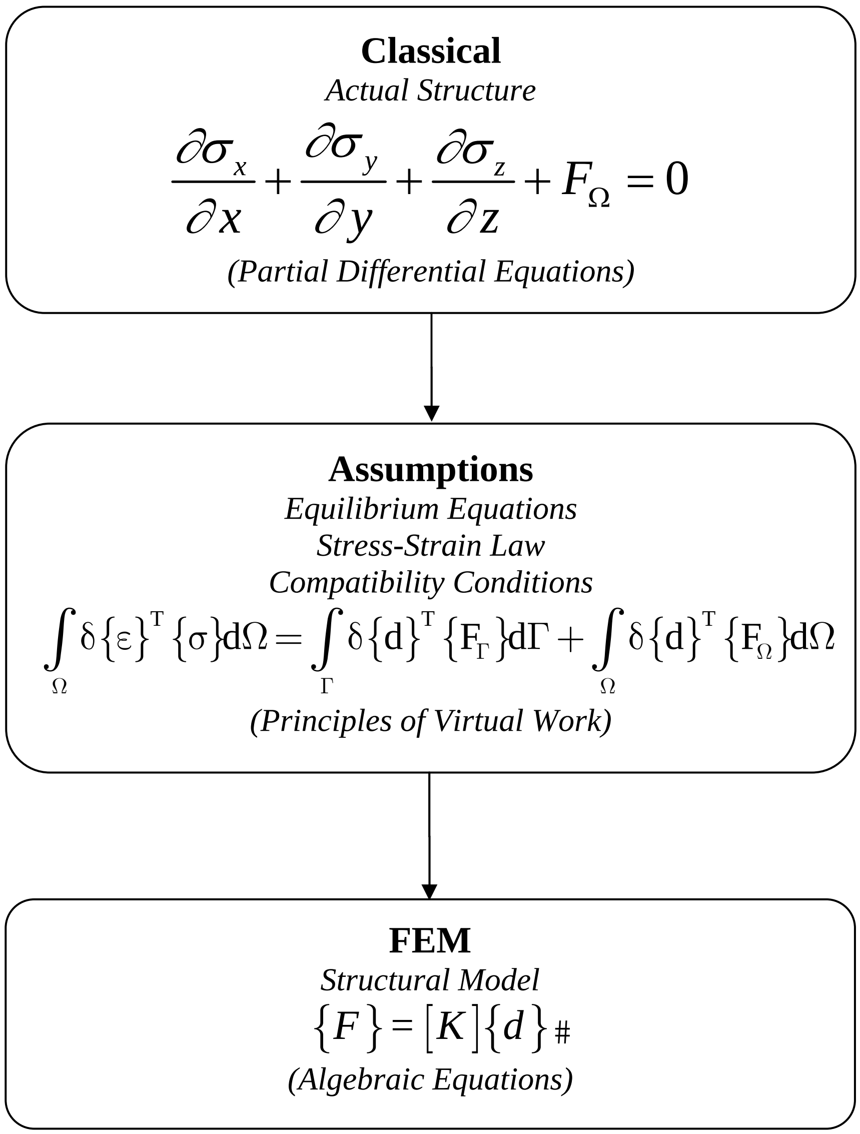


Fig. 1.2.3Fromclassical to FE solution

Varieties of engineering problem like solid and fluid mechanics, heat transfer can easily be solved by the concept of finite element technique. The basic form of the equation will become as follows where action, property and response parameter will vary for case to case as outlined in Table 1.2.1.

$$\{F\} = [K]\{d\} \quad OR \quad \{d\} = [K]^{-1}\{F\}$$

$˝ƗĿı# 3ßƗø˘ß-Ê# 5˘¹øƗı¹˘#

Table 1.2.1 Response parameters for different cases

	Property	**Action**	**Response**
Solid	Stiffness	Load	Displacement
Fluid	Viscosity	Body force	Pressure/Velocity
Thermal	Conductivity	Heat	Temperature

1.3.1 Advantages of FEA

1. The physical properties, which are intractable and complex for any closed bound solution, can be analyzed by this method.
2. It can take care of any geometry (may be regular or irregular).
3. It can take care of any boundary conditions.
4. Material anisotropy and non-homogeneity can be catered without much difficulty.
5. It can take care of any type of loading conditions.
6. This method is superior to other approximate methods like Galerkine and Rayleigh-Ritz methods.
7. In this method approximations are confined to small sub domains.
8. In this method, the admissible functions are valid over the simple domain and have nothing to do with boundary, however simple or complex it may be.
9. Enable to computer programming.

1.3.2 Disadvantages of FEA

1. Computational time involved in the solution of the problem is high.
2. For fluid dynamics problems some other methods of analysis may prove efficient than the FEM.

1.3.3 Limitations of FEA

1. Proper engineering judgment is to be exercised to interpret results.
2. It requires large computer memory and computational time to obtainintend results.
3. There are certain categories of problems where other methods are more effective, e.g., fluid problems having boundaries at infinity are better treated by the boundary element method.
4. For some problems, there may be a considerable amount of input data. Errors may creep up in their preparation and the results thus obtained may also appear to be acceptable which indicates deceptive state of affairs. It is always desirable to make a visual check of the input data.
5. In the FEM,many problems lead to round-off errors. Computer works with a limited number of digits and solving the problem with restricted number of digits may not yield the desired degree of accuracy or it may give total erroneous results in some cases. For many problems the increase in the number of digits for the purpose of calculation improves the accuracy.

1.3.4 Errors and Accuracy in FEA

Every physical problem is formulated by simplifying certain assumptions. Solution to the problem, classical or numerical, is to be viewed within the constraints imposed by these simplifications. The material may be assumed to be homogeneous and isotropic; its behavior may be considered as linearly elastic; the prediction of the exact load in any type of structure is next to impossible. As such the true behavior of the structure is to be viewed with in these constraints and obvious errors creep in engineering calculations.

1. The results will be erroneous if any mistake occurs in the input data. As such, preparation of the input data should be made with great care.
2. When a continuum is discretised, an infinite degrees of freedom system is converted into a model having finite number of degrees of freedom. In a continuum, functions which are continuous are now replaced by ones which are piece-wise continuous within individual elements. Thus the actual continuum is represented by a set of approximations.
3. The accuracy depends to a great extent on the mesh grading of the continuum. In regions of high strain gradient, higher mesh grading is needed whereas in the regions of lower strain, the mesh chosen may be coarser. As the element size decreases, the discretisation error reduces.
4. Improper selection of shape of the element will lead to a considerable error in the solution. Triangle elements in the shape of an equilateral or rectangular element in the shape of a square will always perform better than those having unequal lengths of the sides. For very long shapes, the attainment of convergence is extremely slow.
5. In the finite element analysis, the boundary conditions are imposed at the nodes of the element whereas in an actual continuum, they are defined at the boundaries. Between the

nodes, the actual boundary conditions will depend on the shape functions of the element forming the boundary.

6. Simplification of the boundary is another source of error. The domain may be reduced to the shape of polygon. If the mesh is refined, then the error involved in the discretized boundary may be reduced.
7. During arithmetic operations, the numbers would be constantly round-off to some fixed working length. These round–off errors may go on accumulating and then resulting accuracy of the solution may be greatly impaired.

Steps in Finite Element Analysis

1.4.1 Loading Conditions

There are multiple loading conditions which may be applied to a system. The load may be internal and/or external in nature. Internal stresses/forces and strains/deformations are developed due to the action of loads.Most loads are basically "Volume Loads" generated due to mass contained in a volume. Loads may arise from fluid-structure interaction effects such as hydrodynamic pressure of reservoir on dam, waves on offshore structures, wind load on buildings, pressure distribution on aircraft etc. Again, loads may be static, dynamic or quasi-static in nature. All types of static loads can be represented as:

- Point loads
- Line loads
- Area loads
- Volume loads

The loads which are not acting on the nodal points need to be transferred to the nodes properly using finite element techniques.

1.4.2 Support Conditions

In finite element analysis, support conditions need to be taken care in the stiffness matrix of the structure. For fixed support, the displacement and rotation in all the directionswill be restrained and accordingly, the global stiffness matrix has to modify. If the support prevents translation only in one direction, it can be modeled as 'roller' or 'link supports'. Such link supports are commonly used in finite element software to represent the actual structural state. Sometimes, the support itself undergoes translation under loadings. Such supports are called as 'elastic support' and are modeled with 'spring'. Such situation arises if the structures are resting on soil. The supports may be represented in finite element modeling as:

- Point support
- Line support
- Area support
- Volume support

1.4.3 Type of Engineering Analysis

Finite element analysis consists of linear and non-linear models. On the basis of the structural system and its loadings, the appropriate type of analysis is chosen. The type of analysis to be carried out depends on the following criteria:

- Type of excitation (loads)
- Type of structure (material and geometry)
- Type of response

Considering above aspects, types of engineering analysis are decided. FEA is capable of using multiple materials within the structure such as:

- Isotropic (i.e., identical throughout)
- Orthotropic (i.e., identical at 90^0)
- General anisotropic (i.e., different throughout)

The Equilibrium Equations for different cases are as follows:

1. Linear-Static:

$$Ku = F \tag{1.4.1}$$

2. Linear-Dynamic

$$M\ddot{u}(t) + C\dot{u}(t) + Ku(t) = F(t) \tag{1.4.2}$$

3. **Nonlinear - Static**

$$Ku + F_{NL} = F \tag{1.4.3}$$

1. Nonlinear-Dynamic

$$M\ddot{u}(t) + C\dot{u}(t) + Ku(t) + F(t)_{NL} = F(t) \tag{1.4.4}$$

Here, *M*, *C*, *K*, *F* and *U* are mass, damping, stiffness, force and displacement of the structure respectively. Table 1.4.1 shows various types of analysis which can be performed according to engineering judgment.

Table 1.4.1 Types ofanalysis

Excitation	Structure	Response	Basic analysis type
Static	Elastic	Linear	Linear-Elastic-Static Analysis
Static	Elastic	Nonlinear	Nonlinear-Elastic-Static Analysis
Static	Inelastic	Linear	Linear-Inelastic-Static Analysis
Static	Inelastic	Nonlinear	Nonlinear-Inelastic-Static Analysis
Dynamic	Elastic	Linear	Linear-Elastic-Dynamic Analysis
Dynamic	Elastic	Nonlinear	Nonlinear-Elastic-Dynamic Analysis
Dynamic	Inelastic	Linear	Linear-Inelastic-Dynamic Analysis
Dynamic	Inelastic	Nonlinear	Nonlinear-Inelastic-Dynamic Analysis

1.4.4 Basic Steps in Finite Element Analysis

The following steps are performed for finite element analysis.

1. **Discretisation of the continuum**: The continuum is divided into a number of elements by imaginary lines or surfaces. The interconnected elements may have different sizes and shapes.
2. **Identification of variables:** The elements are assumed to be connected at their intersecting points referred to as nodal points. At each node, unknown displacements are to be prescribed.
3. **Choice of approximating functions:** Displacement function is the starting point of the mathematical analysis. This represents the variation of the displacement within the element. The displacement function may be approximated in the form a linear function or a higher-order function. A convenient way to express it is by polynomial expressions. The shape or geometry of the element may also be approximated.
4. **Formation of the element stiffness matrix**: After continuum is discretised with desired element shapes, the individual element stiffness matrix is formulated. Basically it is a minimization procedure whatever may be the approach adopted. For certain elements, the form involves a great deal of sophistication. The geometry of the element is defined in reference to the global frame. Coordinate transformation must be done for elements where it is necessary.
5. **Formation of overall stiffness matrix**: After the element stiffness matrices in global coordinates are formed, they are assembled to form the overall stiffness matrix. The assembly is done through the nodes which are common to adjacent elements. The overall stiffness matrix is symmetric and banded.

6. **Formation of the element loading matrix:** The loading forms an essential parameter in any structural engineering problem. The loading inside an element is transferred at the nodal points and consistent element matrix is formed.
7. **Formation of the overall loading matrix:** Like the overall stiffness matrix, the element loading matrices are assembled to form the overall loading matrix. This matrix has one column per loading case and it is either a column vector or a rectangular matrix depending on the number of loading cases.
8. **Incorporation of boundary conditions**: The boundary restraint conditions are to be imposed in the stiffness matrix. There are various techniques available to satisfy the boundary conditions. One is the size of the stiffness matrix may be reduced or condensed in its final form. To ease computer programming aspect and to elegantly incorporate the boundary conditions, the size of overall matrix is kept the same.
9. **Solution of simultaneous equations:**The unknown nodal displacements are calculated by the multiplication of force vector with the inverse of stiffness matrix.
10. **Calculation of stresses or stress-resultants:** Nodal displacements are utilized for the calculation of stresses or stress-resultants. This may be done for all elements of the continuum or it may be limited to some predetermined elements. Results may also be obtained by graphical means. It may desirable to plot the contours of the deformed shape of the continuum.

The basic steps for finite element analysis are shown in the form of flow chart below:

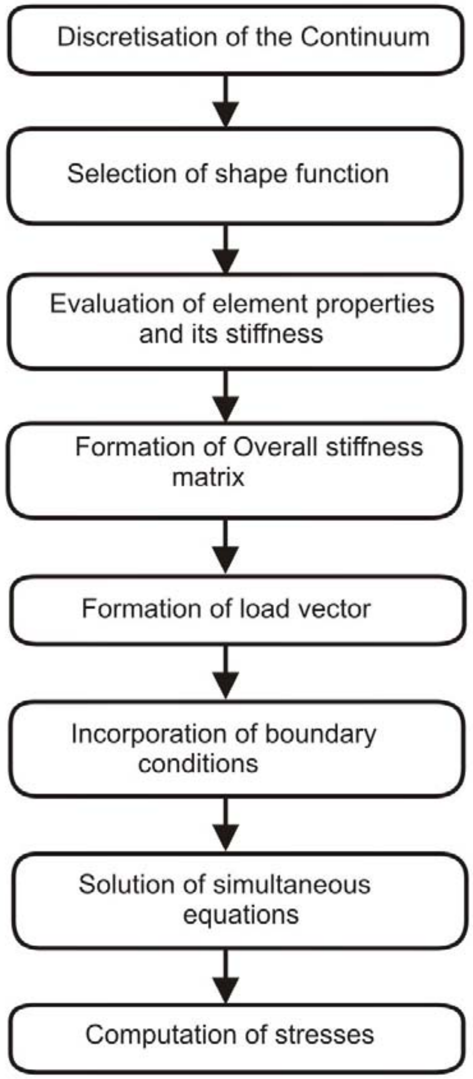


Fig. 1.4.1 Flowchart for steps in FEA

1.4.5 Element Library in FEA Software

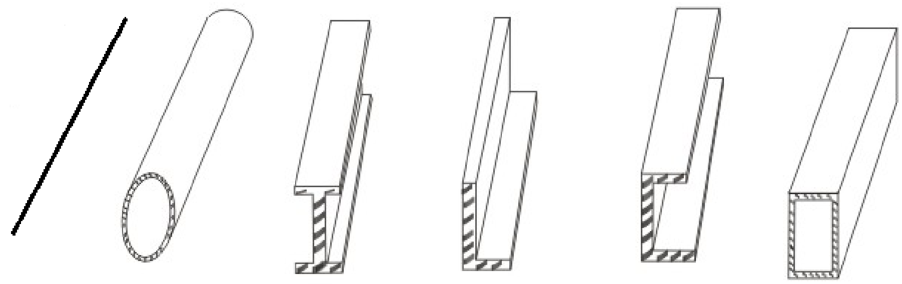
A real structure can be modeled with various ways with appropriate assumptions. The structure may be divided into following categories:

- Cable or tension structures
- Skeletal or framed structures
- Surface or spatial structures
- Solid structures
- Mixed structures

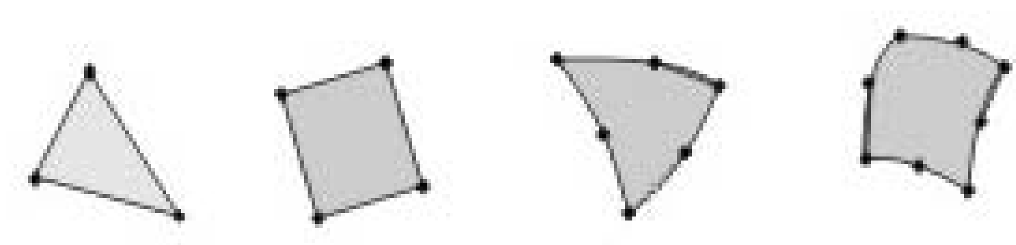
The configuration of structural elements depends upon the geometry of the structural system and the number of independent space coordinates (i.e., *x*, *y* and *z*) required to describe the problem. Thus, the element can be categorized as one, two or three dimensional element. One dimensional element can be represented by a straight line whose ends will be nodal points. The skeletal structures are generally modeled by this type of elements. The pin jointed bar or truss element is the simplest structural element. This element undergoes only axial deformation. The beam element is another type of element which undergoes in-plane transverse displacements and rotations. The frame element is the combination of truss and beam element. Thus, the frame element has axial and in-plane transverse displacements and rotations. This element is generally used to model 1D, 2D and 3D skeletal structural systems. Two-dimensional elements are generally used to model 2D and 3D continuum. These elements are of constant thickness and material properties. The shapes of these elements are triangular or rectangular and it consists of 3 to 9 or even more nodes. These elements are used to solve many problems in solid mechanics such as plane stress, plane strain, plate bending. Three-dimensional element is the most cumbersome which is generally used to model the 3-D continuum. The elements have 6 to 27 numbers of nodes or more. Because of large degrees of freedom, the analysis is time consuming using 3-D elementsand difficult to interpret its results. However, for accurate analysis of the irregular continuum, 3-D elements are useful. To analyze any real structure, appropriate elements are to be assigned for the finite element analysis. In standard FEA software, following types of element library are used to discretize the domain.

- Truss element
- Beam element
- Frame element
- Membrane/ Plate/Shell element
- Solid element
- Composite element
- Shear panel
- Spring element
- Rigid/Link element
- Viscous damping element

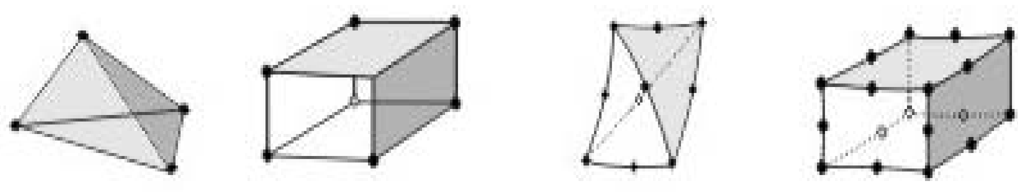
The different types of elements available in standard finite element software are shown in Fig. 1.4.2.



1D Elements (Truss,beam, grid and frame)



2D Elements(Plane stress, Plane strain, Axisymmetric, Plate and Shell)



3D Elements

Fig. 1.4.2Varioustypes of elements for computer modeling

Finite Element Formulation Techniques
Virtual Work and Variational Principle

1.5.1 Introduction

Finite element formulation can be constructed from governing differential equations over a domain. This can be formulated by various ways like Virtual Work Method, VariationalMethod, Weighted Residual Method etc.

1.5.2 Principle of Virtual Work

The principle of virtual work is a very useful approach for solving varieties of structural mechanics problem. When the force and displacement are unrelated to the cause and effect relation, the work is called virtual work. Therefore, the virtual work may be caused by true force moving through imaginary displacements or vice versa. Thus, the principle of virtual work can be divided into two categories: (a) principle of virtual forces and (b) principle of virtual displacements. The principle of virtual forces establishes the compatibility conditions. The principle of virtual displacements establishes the conditions of equilibrium and is used in the displacement model of the finite element technique.

The external virtual work is the work done by real load moving through imaginary displacements in a structure. These loads include both the load distributed over the entire surface and volume. Thus, the virtual work done by the external force is:

$$\delta W_E = \int_{\Gamma} \{\delta u \quad \delta v \quad \delta w\} \begin{Bmatrix} F_{\Gamma x} \\ F_{\Gamma y} \\ F_{\Gamma z} \end{Bmatrix} d\Gamma + \int_{\Omega} \{\delta u \quad \delta v \quad \delta w\} \begin{Bmatrix} F_{\Omega x} \\ F_{\Omega y} \\ F_{\Omega z} \end{Bmatrix} d\Omega \tag{1.5.1}$$

Where, δu, δv and δw are the components of the virtual displacements in x, y and z direction respectively. $F_{\Gamma x}$, $F_{\Gamma y}$ and $F_{\Gamma z}$ are the surface forces and $F_{\Omega x}$, $F_{\Omega y}$and $F_{\Omega z}$ are the body forces in x, y and z direction respectively. In the above equation, the integration is carried out over the entire surface in the first term and over the entire volume in the second term. The above expression can be rewritten as:

$$\delta W_E = \int_{\Gamma} \delta\{d\}^T \{F_{\Gamma}\} d\Gamma + \int_{\Omega} \delta\{d\}^T \{F_{\Omega}\} d\Omega \tag{1.5.2}$$

Here, $\{d\}^T = \{u \quad v \quad w\}$.For the three dimensional stress-strain condition, there are six components of stresses $(\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{zx})$ and six components of strains in virtual displacement fields $(\delta\varepsilon_x, \delta\varepsilon_y, \delta\varepsilon_z, \delta\gamma_{xy}, \delta\gamma_{yz}, \delta\gamma_{zx})$.Therefore, the virtual internal work can be expressed as follows:

$$\delta U = \int_{\Omega} \{\delta\varepsilon_x \quad \delta\varepsilon_y \quad \delta\varepsilon_z \quad \delta\gamma_{xy} \quad \delta\gamma_{yz} \quad \delta\gamma_{zx}\} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} d\Omega \qquad (1.5.3)$$

Or

$$\delta U = \int_{\Omega} \delta\{\varepsilon\}^T \{\sigma\} d\Omega \qquad (1.5.4)$$

According to principle of virtual work, the work done by external forces due to the virtual displacement of a structure in equilibrium is equal to the work done by the internal forces for the virtual internal displacement. Therefore, $\delta W_E = \delta U$ Thus eqs. (1.5.2) and (1.5.4) can be made equal and can be related as follows:

$$\int_{\Gamma} \delta\{d\}^T \{F_\Gamma\} d\Gamma + \int_{\Omega} \delta\{d\}^T \{F_\Omega\} d\Omega = \int_{\Omega} \delta\{\varepsilon\}^T \{\sigma\} d\Omega \qquad (1.5.5)$$

1.5.3 Variational Principle

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

$$U = \frac{1}{2} \iiint_{\Omega} \{\varepsilon\}^T \{\sigma\} d\Omega \qquad (1.5.6)$$

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

$$U = \frac{1}{2} \iiint_{\Omega} [[B]\{d\}]^T [D]\{B\}\{d\} d\Omega \qquad (1.5.7)$$

Applying the variational principle one can express

$$\{F\} = \frac{\partial U}{\partial \{d\}} = \iiint_{\Omega} [B]^T [D][B] d\Omega \{d\} \tag{1.5.8}$$

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

$$[K] = \iiint_{\Omega} [B]^T [D][B] d\Omega \tag{1.5.9}$$

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

1.5.4Weighted ResidualMethod

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

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

The mathematical statement of a physical problem can be defined as:

In domainΩ,

$$\mathrm{D}u - f = 0 \tag{1.5.10}$$

Where,

D is the differential operator

u = u(x) = dependent variables such as displacement, pressure, velocity, potential function

x = independent variables such as coordinates of a point

f = a function of x which may be constant or zero

If $\bar{u}$ is an approximate solution then residual in domainΩ,

$$R = \mathrm{D}\bar{u} - f \tag{1.5.11}$$

According to the weighted residual method, the weak form of above equation will become

$$\int_{\Omega} w_i \; R \; d\Omega = 0 \quad \text{for } i = 1, 2, 3, \ldots, n$$

or (1.5.12)

$$\int_{\Omega} w_i \left(D\bar{u} - f \right) d\Omega = 0$$

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

Galerkin Method

1.6.1 Introduction

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

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

For example, in case of a beam problem ($EI\frac{\partial^4 y}{\partial x^4} - q = 0$) differential equation is of fourth order. As a result, displacement and slope will be essential boundary condition where as moment and shear will be non-essential boundary condition.

1.6.2 Galerkin Method for2D Elasticity Problem

For a two dimensional elasticity problem, equation of equilibrium can be expressed as

$$\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + F_{\Omega x} = 0 \qquad (1.6.1)$$

$$\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + F_{\Omega y} = 0 \qquad (1.6.2)$$

Where, $F_{\Omega x}$ and $F_{\Omega y}$ are the body forces in X and Y direction respectively. Let assume, $\Gamma_{\Gamma x}$ and $\Gamma_{\Gamma y}$ are surface forces in X and Y direction and α as angle made by normal to surface with X– axis (Fig. 2.2.1). Therefore, force equilibrium of element can be written as:

$$F_{\Gamma x}(PQ)t = \sigma_x (OP)t + \tau_{xy}(OQ)t$$

$$F_{\Gamma x} = \sigma_x \frac{OP}{PQ} + \tau_{xy}\frac{OQ}{PQ} = \sigma_x \cos\alpha + \tau_{xy}\sin\alpha = \sigma_x \cos\alpha + \tau_{xy} Cos(90-\alpha)$$

$$\text{Thus, } F_{\Gamma x} = \sigma_x \ell + \tau_{xy} m \qquad (1.6.3)$$

Where, ℓ and m are direction cosines of normal to the surface. Similarly,

$$F_{\Gamma y} = \tau_{xy}\ell + \sigma_y m \qquad (1.6.4)$$

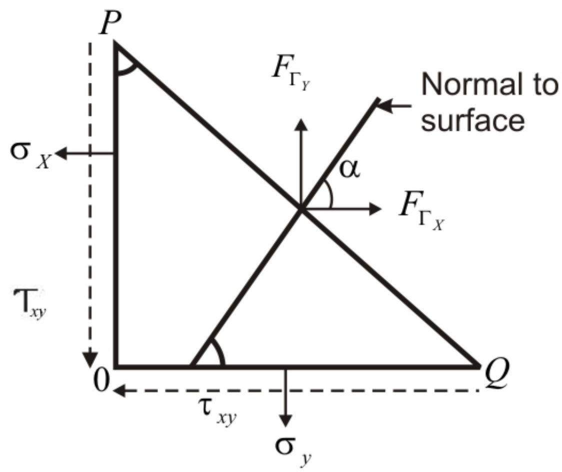


Fig. 1.6.1 Elemental stresses in 2D

AdoptingGalerkin'sapproach using eq. (1.6.2 and 1.6.3)

$$\left[\iint\left(\frac{\partial\sigma_x}{\partial x}+\frac{\partial\tau_{xy}}{\partial y}+F_{\Gamma x}\right)\delta u+\iint\left(\frac{\partial\tau_{xy}}{\partial x}+\frac{\partial\sigma_y}{\partial y}+F_{\Gamma y}\right)\delta v\right]dxdy=0 \qquad (1.6.5)$$

Where δu and δv are weighting functions i.e elemental displacements in X and Y directions respectively. Now one can expand above equation by using Green's Theorem.

Green Theorem states that if $\phi(x,y)$ and $\psi(x,y)$ are continuous functions then their first and second partial derivatives are also continuous. Therefore,

$$\iint\left[\frac{\partial\phi}{\partial x}\frac{\partial\psi}{\partial x}+\frac{\partial\phi}{\partial y}\frac{\partial\psi}{\partial y}\right]dxdy=-\iint\phi\left[\frac{\partial^2\psi}{\partial x^2}+\frac{\partial^2\psi}{\partial y^2}\right]dxdy+\int\phi\left[\frac{\partial\psi}{\partial x}\ell+\frac{\partial\psi}{\partial y}m\right]ds \qquad (1.6.6)$$

Assuming, $\phi=\sigma_x;\frac{\partial\psi}{\partial x}=\delta u;\ \frac{\partial\psi}{\partial y}=0$ one can rewrite with the use of above relationas

$$\iint\frac{\partial\sigma_x}{\partial x}\delta u\ dx\ dy=-\iint\sigma_x\frac{\partial(\delta u)}{\partial x}dx\ dy+\int\sigma_x\ell\ \delta u\ ds \qquad (1.6.7)$$

Similarly, assuming $\phi=\sigma_y;\frac{\partial\psi}{\partial x}=0$ and $\frac{\partial\psi}{\partial y}=\delta v$

$$\iint\frac{\partial\sigma_y}{\partial y}\delta v\ dx\ dy=-\iint\sigma_y\frac{\partial(\delta v)}{\partial y}dx\ dy+\int\sigma_y m\ \delta v\ ds \qquad (1.6.8)$$

Again, assuming $\phi = \tau_{xy}; \dfrac{\partial \psi}{\partial x} = \delta v; \dfrac{\partial \psi}{\partial y} = 0$

$$\iint \frac{\partial \tau_{xy}}{\partial y} \delta v \, dx \, dy = -\iint \tau_{xy} \frac{\partial (\delta v)}{\partial x} dx \, dy + \int \tau_{xy} \ell \, \delta v \, ds \tag{1.6.9}$$

And assuming, $\phi = \tau_{xy}; \dfrac{\partial \psi}{\partial x} = 0; \dfrac{\partial \psi}{\partial y} = \delta u$

$$\iint \frac{\partial \tau_{xy}}{\partial y} \delta u \, dx \, dy = -\iint \tau_{xy} \frac{\partial (\delta u)}{\partial y} dx \, dy + \int \tau_{xy} m \, \delta u \, ds$$

Putting values of eqs.(2.2.7), (2.2.8) and (2.2.9), in eq. (2.2.5), one can get the following relation:

$$-\iint \left[\sigma_x \frac{\partial}{\partial x}(\delta u) + \sigma_y \frac{\partial}{\partial y}(\delta v) + \tau_{xy} \frac{\partial}{\partial x}(\delta v) + \tau_{xy} \frac{\partial}{\partial y}(\delta u) \right] dx \, dy$$
$$+ \int \left[\sigma_x \ell \delta u + \sigma_y m \delta v + \tau_{xy} \ell \delta v + \tau_{xy} m \delta u \right] ds + \iint F_{\Omega x} \delta u \, dx \, dy + \iint F_{\Omega y} \delta v \, dx \, dy = 0 \tag{1.6.10}$$

Rearranging the terms of above expression, the following relations are obtained.

$$-\iint \left[\sigma_x \frac{\partial}{\partial x}(\delta u) + \sigma_y \frac{\partial}{\partial y}(\delta v) + \tau_{xy} \frac{\partial}{\partial x}(\delta v) + \tau_{xy} \frac{\partial}{\partial y}(\delta u) \right] dx \, dy + \iint \left(F_{\Omega x} \delta u + F_{\Omega y} \delta v \right) dx \, dy$$
$$+ \int \left(\sigma_x \ell + \tau_{xy} m \right) \delta u ds + \int \left(\tau_{xy} \ell + \sigma_y m \right) \delta v ds = 0 \tag{1.6.11}$$

Here, $F_{\Omega x}$ and $F_{\Omega y}$ are the body forces and δu & δv are virtual displacements in X and Ydirections respectively.

Considering firstterm of eq. (2.2.11), virtual displacement δu is given to the element of unit thickness. Dotted position in Fig. 2.2.2 shows the virtual displacement. Thus, work done by σ_x :

$$\sigma_x dy \left[\delta u + \frac{\partial}{\partial x}(\delta u) dx \right] - \sigma_x dy \delta u = \sigma_x \frac{\partial}{\partial x}(\delta u) dx dy \tag{1.6.12}$$

Similarly, considering secondterm of eq. (1.6.11), virtual work done by body forces is

$$\iint\left(F_{\Omega x}\delta u+F_{\Omega y}\delta v\right)dx\ dy$$

Putting eqs.(1.6.3) &(1.6.4) in third term of eq. (1.6.11) we get the virtual work done by surface forces as:

$$\int F_{\Gamma x}\delta u ds+\int F_{\Gamma y}\delta v ds$$

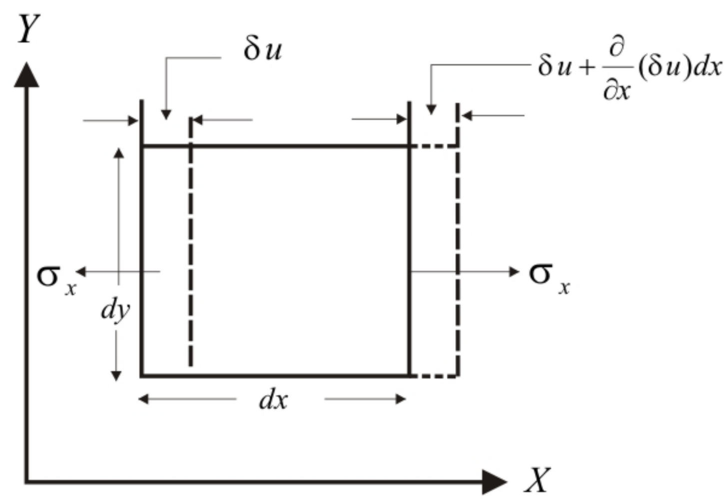


Fig. 2.2.2 Element subjected to stresses

Due to virtual displacement δu, change in strain $\delta\in_x$ is given by:

$$\delta\in_x=\frac{\left[\delta u+\frac{\partial}{\partial x}(\delta u)dx\right]-\delta u}{dx}=\frac{\partial}{\partial x}(\delta u) \tag{1.6.13}$$

The virtual work doneby σ_x is $\sigma_x.\delta\in_x.dxdy$. Similarly all the individual term in the first term of eq. (1.6.11) can be derived from eq. (1.6.13) which will be as follows:

$$\begin{aligned}
&\iint\sigma_x\frac{\partial}{\partial x}(\delta u)dxdy=\iint\sigma_x\delta\in_x dxdy\\
&\iint\sigma_y\frac{\partial}{\partial y}(\delta v)dxdy=\iint\sigma_y\delta\in_y dxdy\\
&\iint\tau_{xy}\left\{\frac{\partial}{\partial x}(\delta v)+\frac{\partial}{\partial y}(\delta u)\right\}=\iint\tau_{xy}\delta\gamma_{xy}dxdy
\end{aligned} \tag{1.6.14}$$

Now, the work done by internal forces will be

$$\delta U = \iint \left(\sigma_x \delta \in_x + \sigma_y \delta \in_y + \tau_{xy} \delta \gamma_{xy} \right) dxdy \tag{1.6.15}$$

If external work done is represented by W_E and U is the internal work done then,

$$-\delta U + \delta w_E = 0 \text{ or } \delta U = \delta w_E \tag{1.6.16}$$

Thus in elasticity problems, Galerkin's method turns out to be the principle of virtual work, which can be stated that "A Deformable body is said to be in equilibrium,if the total work done by external forces is equal to the total work done by internal forces." The work done above is virtual as either forces or deformations are also virtual. Thus, Galerkin'sapproach can be followed in all problems involving solution of a set of equations subjected to specified boundary values.

1.6.3 Galerkin Method for 2D Fluid Flow Problem

Let consider the two dimensional incompressible fluid equation which can be expressed by pressure variable only as follows.

$$\nabla^2 p = 0 \tag{1.6.17}$$

Where p is the pressure inside the fluid domain. The above equation can be expressed in 2D form as:

$$\frac{\partial^2 p}{\partial x^2} + \frac{\partial^2 p}{\partial y^2} = 0$$

$$\text{or} \tag{1.6.18}$$

$$p,_{ii} = 0$$

Applying weighted residual method, the weak form of the above equation will become

$$\int_\Omega w_i \; p,_{ii} \; d\Omega = 0 \tag{1.6.19}$$

Integrating by parts of the above expression, the following relation can be obtained.

$$\int_\Gamma w_i \; p,_i \; d\Gamma - \int_\Omega w_{i,i} \; p,_i \; d\,\Omega = 0$$

$$\text{or} \int_\Omega w_{i,i} \; p,_i \; d\,\Omega = \int_\Gamma w_i \; p,_i \; d\Gamma \tag{1.6.20}$$

If the nodal pressure and interpolation functions are denoted by $\overline{p}$ and N respectively, then the pressure at any point inside the fluid domain can be expressed as

$$p = [N]\{\overline{p}\}$$

Similarly, the weighted function can also be written with the help of interpolation function as

$$w = [N]\{\overline{w}\}$$

Thus, $p_{i,i} = [L]\{p\} = [L][N]\{\overline{p}\} = [B]\{\overline{p}\}$, where, $[L] = \left[\frac{\partial}{\partial x}\ \frac{\partial}{\partial y}\right]$ = differential operator.

Similarly, $w_{i,i} = [L]\{W\} = [L][N]\{\overline{w}\} = [B]\{\overline{w}\}$

$$\text{Thus,} \int_{\Omega} w_{i,i}\ p_{,i}\ d\,\Omega = \int [\overline{w}]^T [B]^T [B][\overline{p}] d\Omega \tag{1.6.21}$$

$$\int_{\Gamma} w_i\ p_{,i}\ d\,\Gamma = \int_{\Gamma} \{\overline{w}\}^T [N]^T \frac{\partial p}{\partial n} d\Gamma \tag{1.6.22}$$

Here, Γdenotes the surface of the fluid domain and *n* represents the direction normal to the surface. Thus, from eq. (1.6.20), one can write the expression as:

$$\text{Thus,} \int_{\Omega} \{\overline{w}\}^T [B]^T [B]\{\overline{p}\} d\Omega = \int_{\Gamma} \{\overline{w}\}^T [N]^T \frac{\partial p}{\partial n} d\Gamma$$

$$\text{Or,}\quad [G]\{\overline{p}\} = \{S\} \tag{1.6.23}$$

Where,

$$[G] = \int_{\Omega} [B]^T [B] d\Omega = \int_{\Omega} \left(\frac{\partial}{\partial x}[N]^T \frac{\partial}{\partial x}[N] + \frac{\partial}{\partial y}[N]^T \frac{\partial}{\partial y}[N] \right) d\Omega$$

$$\text{and}\ \{S\} = \int_{\Gamma} [N]^T \frac{\partial p}{\partial n} d\Gamma \tag{1.6.24}$$

Here, *n* is the direction normal to the surface. Thus, solving the above equationwith the prescribed boundary conditions, one can find out the pressure distribution inside the fluid domain by the use of finite element technique.

Stiffness Matrix and Boundary Conditions

1.7.1Element Stiffness Matrix

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

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

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

1.7.2Global Stiffness Matrix

A structural system is an assemblage of number of elements. These elements are interconnected together to form the whole structure. Therefore, the element stiffness of all the elementsarefirst need to be calculated and then assembled together in systematic manner. It may be noted that the stiffness at a joint is obtained by adding the stiffness of all elements meeting at that joint.

To start with, the degrees of freedom of the structure are numberedfirst. This numbering will start from 1 to n where n is the total degrees of freedom. These numberings are referred to as degrees of freedom corresponding to global degrees of freedom. The element stiffness matrix of each element should be placed in their proper position in the overall stiffness matrix. The following steps may be performed to calculate the global stiffness matrix of the whole structure.

a. Initialize global stiffness matrix $[K]$ as zero. The size of global stiffness matrix will be equal to the total degrees of freedom of the structure.
b. Compute individual element properties and calculate local stiffness matrix $[k]$ of that element.
c. Add local stiffness matrix$[k]$to global stiffness matrix$[K]$ using proper locations
d. Repeat the Step b. and c. till all local stiffness matrices are placed globally.

The steps to be followed in the computer program are shown in the form of flow chart in Fig. 1.7.1 for assembling the local stiffness matrix to global stiffness matrix.

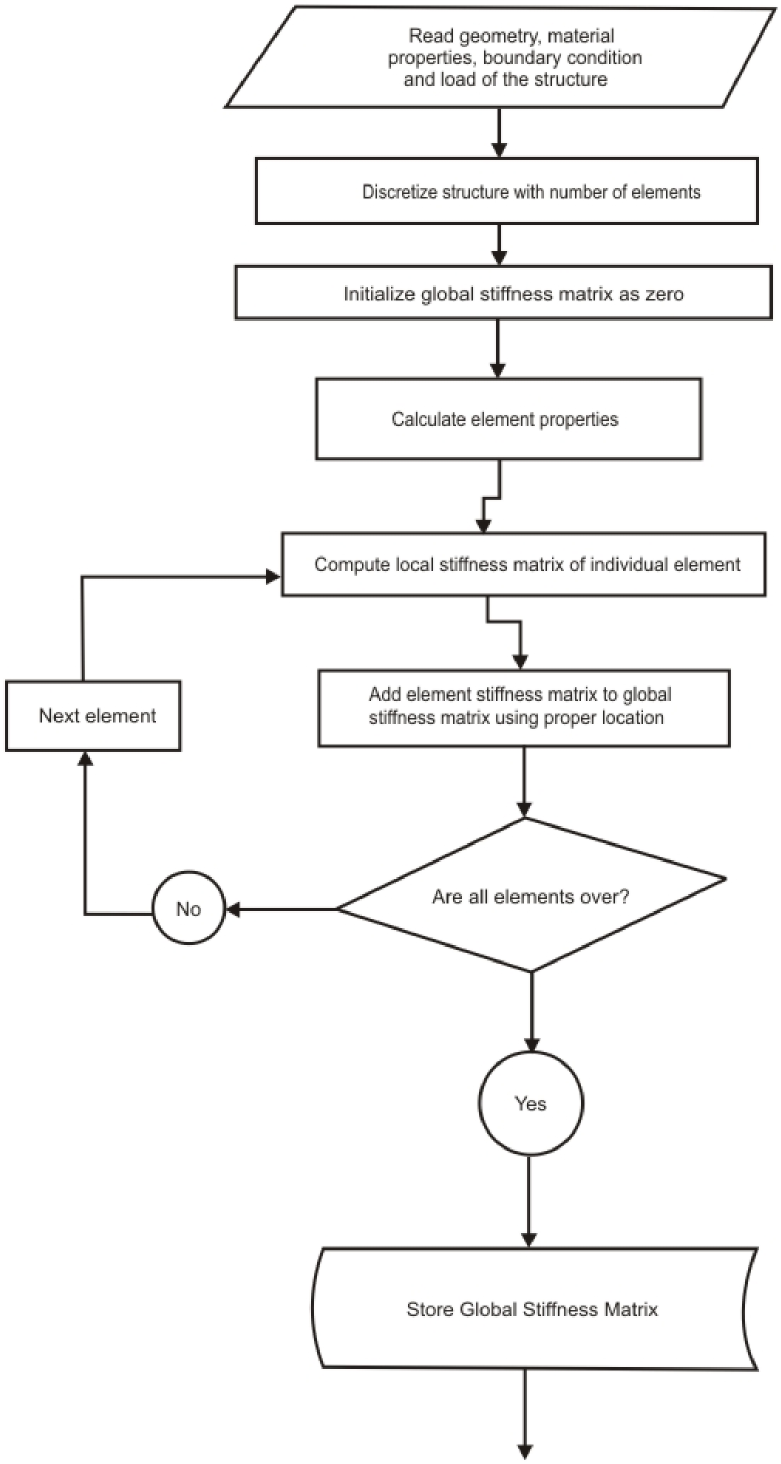


Fig. 1.7.1Assembly of stiffness matrix from local to global

1.7.3Boundary Conditions

Under this section, procedure to include the effect of boundarycondition in the stiffness matrix for the finite element analysis will be discussed. The solution cannot be obtained unless support conditions are included in the stiffness matrix. This is because, if all the nodes of the structure are included in displacement vector, the stiffness matrix becomes singular and cannot be solved if the structure is not supported amply, and it cannot resist the applied loads.A solution cannot be achieved until the boundary conditions *i.e.*, the known displacements are introduced.

In finite element analysis, the partitioning of the global matrix is carried out in a systematic way for the hand calculation as well as for the development of computer codes. In partitioning, normally the equilibrium equations can be partitioned by rearranging corresponding rows and columns, so that prescribed displacements are grouped together. For example, let considerthe equation of equilibrium is expressed in compact form as:

$$\{F\} = [K]\{d\} \tag{1.7.1}$$

Where,

$[K]$ is the global stiffness matrix,

$\{d\}$ is the displacement vector consisting of global degrees of freedom, and

$\{F\}$ is the load vector corresponding to degrees of freedom.

By the method of partitioning the above equation can be partitioned in the following manner.

$$\begin{Bmatrix} \{F_\alpha\} \\ \{F_\beta\} \end{Bmatrix} = \begin{bmatrix} [K_{\alpha\alpha}] & [K_{\alpha\beta}] \\ [K_{\beta\alpha}] & [K_{\beta\beta}] \end{bmatrix} \begin{Bmatrix} \{d_\alpha\} \\ \{d_\beta\} \end{Bmatrix} \tag{1.7.2}$$

Where,subscripts α refers to the displacements free to move andβ refers to the prescribed support displacements.As the prescribed displacements {d } are known,eq. (1.7.2) may be written in expanded form as:

$$\{F_\alpha\} = [K_{\alpha\alpha}]\{d_\alpha\} + [K_{\alpha\beta}]\{d_\beta\} \tag{1.7.3}$$

Thus it is possible to obtain the free displacement of the structure$\{d_\alpha\}$ as

$$\{d_\alpha\} = [K_{\alpha\alpha}]^{-1}\{\{F_\alpha\} - [K_{\alpha\beta}]\{d_\beta\}\} \tag{1.7.4}$$

If the displacements at supports $\{d_\beta\}$are zero, then the above equation can be simplified to the following expression.

$$\{d_\alpha\} = [K_{\alpha\alpha}]^{-1}\{F_\alpha\} \tag{1.7.5}$$

Thus, by rearranging assembled matrix, the portion corresponding to the unknown displacements in eq.(2.4.4) can be taken out for the solution purpose. This is possible as the known displacements $\{d_\beta\}$are restrained, i.e., displacementsare zero. If the support has some known displacements, then eq. (2.4.4) can be used to find the solution. If the few supports of the structures yield, then the above method may be modified by partitioning the stiffness matrix into three parts as shown below:

$$\begin{Bmatrix} \{F_\alpha\} \\ \{F_\beta\} \\ \{F_\gamma\} \end{Bmatrix} = \begin{bmatrix} [K_{\alpha\alpha}] & [K_{\alpha\beta}] & [K_{\alpha\gamma}] \\ [K_{\beta\alpha}] & [K_{\beta\beta}] & [K_{\beta\gamma}] \\ [K_{\gamma\alpha}] & [K_{\gamma\beta}] & [K_{\gamma\gamma}] \end{bmatrix} \begin{Bmatrix} \{d_\alpha\} \\ \{d_\beta\} \\ \{d_\gamma\} \end{Bmatrix} \tag{1.7.6}$$

Here, α refers to unknown displacement; β refers to known displacement ($\neq 0$) and γ refers to zero displacement. Thus, the above equation can be separated and solved independently to find required unknown results as shown below.

$$\{F_\alpha\} = [K_{\alpha\alpha}]\{d_\alpha\} + [K_{\alpha\beta}]\{d_\beta\} + [K_{\alpha\gamma}]\{d_\gamma\}$$

$$\text{or, } [K_{\alpha\alpha}]\{d_\alpha\} = \{F_\alpha\} - [K_{\alpha\beta}]\{d_\beta\} \quad \text{as } \{d_\gamma\} = \{0\}$$

$$\text{Thus, } \{d_\alpha\} = [K_{\alpha\alpha}]^{-1}\{\{F_\alpha\} - [K_{\alpha\beta}]\{d_\beta\}\} \tag{1.7.7}$$

For computer programming, several techniques are available for handling boundary conditions. One of the approachesis to make the diagonal element of stiffness matrix corresponding to zero displacement as unity and corresponding all off-diagonal elements as zero. For example,let consider a 3×3 stiffness matrix with following force-displacement relationship.

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} \\ k_{21} & k_{22} & k_{23} \\ k_{31} & k_{32} & k_{33} \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \end{Bmatrix} \tag{1.7.8}$$

Now, if the third node has zero displacement (i.e., $d_3 = 0$) then the matrix will be modified as follows to incorporate the boundary condition.

$$\begin{Bmatrix} F_1 \\ F_2 \\ 0 \end{Bmatrix} = \begin{bmatrix} k_{11} & k_{12} & 0 \\ k_{21} & k_{22} & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \end{Bmatrix} \tag{1.7.9}$$

Thus, while inverting whole matrix, d_3 will become zero automatically.

To incorporate known support displacement in computer programming following procedure may be adopted. Considering the displacement $d_2$has known value of δ, 1st row of eq. (2.4.8) can be written as:

$$F_1 = k_{11} \times d_1 + k_{12} \times d_2 + k_{13} \times d_3 \tag{1.7.10}$$

Or

$$F_1 - k_{12} \times \delta = k_{11} \times d_1 + k_{13} \times d_3 \tag{1.7.11}$$

Now the 2nd row of eq. (2.4.8) has to become:

$$\{\delta\} = \{d_2\} \tag{1.7.12}$$

Similarly 3rd row will be:

$$F_3 - k_{32} \times \delta = k_{31} \times d_1 + k_{33} \times d_3 \tag{1.7.13}$$

Thus above three equations can be written in a combined form as

$$\begin{Bmatrix} F_1 - k_{12}\delta \\ \delta \\ F_3 - k_{32}\delta \end{Bmatrix} = \begin{bmatrix} k_{11} & 0 & k_{13} \\ 0 & 1 & 0 \\ k_{31} & 0 & k_{33} \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \end{Bmatrix} \tag{1.7.14}$$

Another approach may also be followed to take care the known restrained displacementsby assigning a higher value δ(say $\delta = 10^{20}$) in the diagonal element corresponding to that displacement.

$$\begin{Bmatrix} F_1 \\ \delta \times 10^{20} \times k_{22} \\ F_3 \end{Bmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} \\ k_{21} & k_{22} \times 10^{20} & k_{23} \\ k_{31} & k_{32} & k_{33} \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \end{Bmatrix} \tag{1.7.15}$$

$$\therefore \delta \times 10^{20} \times k_{22} = k_{21} d_1 + k_{22} \times 10^{20} \times d_2 + k_{23} \times d_3$$

As d_3 is corresponding to zero displacement, the above equation can be simplified to the following.

$$\therefore \delta \times 10^{20} \times k_{22} = k_{21} d_1 + k_{22} \times 10^{20} \times d_2$$

$$\text{or } \delta \times 10^{20} \times k_{22} = k_{22} \times 10^{20} \times d_2$$

$$\Rightarrow d_2 = \delta \rightarrow \text{known displacement is ensured}$$

If the overall stiffness matrix is to be formed in half band form then the numbering of nodes should be such that the bandwidth is minimum. For this the labels are put in a systematic manner irrespective of whether the joint displacements are unknowns or restraints. However, if the unknown displacements are labeled first then the matrix operations can be restricted up to unknown displacement labels and beyond that the overall stiffness matrix may be ignored.

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

SCHOOL OF MECHANICAL ENGINEERING
DEPARTMENT OF AERONAUTICAL ENGINEERING

UNIT – II –FINITE ELEMENT METHOD FOR AIRCRAFT STRUCTURES – SAEA1504

UNIT – II

BEAM BENDING

1. Introduction

A beam is a structural member which is capable of withstanding load primarily by resisting bending. The primary tool for analysis of beam is the Euler–Bernoulli beam equation. Other methods for determining the deflection of beams include "slope deflection method" and "method of virtual work". For calculation of internal forces of beam include "moment distribution method", force or flexibility method and stiffness method. However, all these methods have limitations if either of geometry, loading, material properties or boundary conditions becomes arbitrary in nature. Finite element techniques can well handle such cases and relieve the analyzer of making simplifications to arrive approximate solutions.

Types of Beams

The following are the important types of beams.

(a) Cantilever Beam
(b) Simply Supported Beam
(c) Overhanging Beam
(d) Fixed Beam
(e) Continuous Beam

1.1 Derivation of Shape Function:

The degrees of freedom at each node for a beam member will be (i) vertical deflection and (ii) rotation. For a beam member, the slope of the elastic curve θ is given by: $\theta = \frac{dv}{dx}$, where the variable v is the displacement function of the beam. As the beam has two degrees of freedom at each node, the variation of v will be cubic and can be expressed using Pascal's triangle as:

$$v(x) = \alpha_0 + \alpha_1 x + \alpha_2 x^2 + \alpha_3 x^3 = \begin{bmatrix} 1 & x & x^2 & x^3 \end{bmatrix} \begin{Bmatrix} \alpha_0 \\ \alpha_1 \\ \alpha_2 \\ \alpha_3 \end{Bmatrix}$$

……. (2.1)

and

$$\theta = \frac{dv}{dx} = \begin{bmatrix} 0 & 1 & 2x & 3x^2 \end{bmatrix} \begin{Bmatrix} \alpha_0 \\ \alpha_1 \\ \alpha_2 \\ \alpha_3 \end{Bmatrix}$$

……. (2.2)

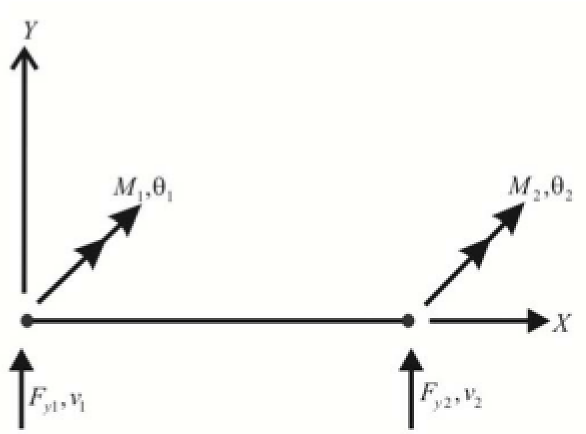


Fig. 2.1 Beam element

Now, applying boundary conditions, the following expressions from the above relations can be obtained:

At x=0:

$$V_1 = \begin{bmatrix} 1 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} \alpha_0 \\ \alpha_1 \\ \alpha_2 \\ \alpha_3 \end{Bmatrix}; \ \theta_1 = \begin{bmatrix} 0 & 1 & 0 & 0 \end{bmatrix} \begin{Bmatrix} \alpha_0 \\ \alpha_1 \\ \alpha_2 \\ \alpha_3 \end{Bmatrix};$$

At $x=L$:

$$V_2 = \begin{bmatrix} 1 & L & L^2 & L^3 \end{bmatrix} \begin{Bmatrix} \alpha_0 \\ \alpha_1 \\ \alpha_2 \\ \alpha_3 \end{Bmatrix}; \ \theta_2 = \begin{bmatrix} 0 & 1 & 2L & 3L^2 \end{bmatrix} \begin{Bmatrix} \alpha_0 \\ \alpha_1 \\ \alpha_2 \\ \alpha_3 \end{Bmatrix}$$

\

Thus combining the above expressions one can write:

$$\begin{Bmatrix} V_1 \\ \theta_1 \\ V_2 \\ \theta_2 \end{Bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 1 & L & L^2 & L^3 \\ 0 & 1 & 2L & 3L^2 \end{bmatrix} \begin{Bmatrix} \alpha_0 \\ \alpha_1 \\ \alpha_2 \\ \alpha_3 \end{Bmatrix} = [A]\{\alpha\}$$

……..(2.3)

So,

$$\begin{Bmatrix} \alpha_0 \\ \alpha_1 \\ \alpha_2 \\ \alpha_3 \end{Bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 1 & L & L^2 & L^3 \\ 0 & 1 & 2L & 3L^2 \end{bmatrix}^{-1} \begin{Bmatrix} V_1 \\ \theta_2 \\ V_2 \\ \theta_2 \end{Bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ -\frac{3}{L^2} & -\frac{2}{L} & \frac{3}{L^2} & -\frac{1}{L} \\ \frac{2}{L^3} & \frac{1}{L^2} & -\frac{2}{L^3} & \frac{1}{L^2} \end{bmatrix} \begin{Bmatrix} V_1 \\ \theta_1 \\ V_2 \\ \theta_2 \end{Bmatrix}$$

…….(2.4)

Therefore,

$$v(x) = \begin{bmatrix} 1 & x & x^2 & x^3 \end{bmatrix} \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ -\frac{3}{l^2} & -\frac{2}{l} & \frac{3}{l^2} & -\frac{1}{l} \\ \frac{2}{l^3} & \frac{1}{l^2} & -\frac{2}{l^3} & \frac{1}{l^2} \end{bmatrix} \begin{Bmatrix} V_1 \\ \theta_1 \\ V_2 \\ \theta_2 \end{Bmatrix} = \begin{bmatrix} N_1 & N_2 & N_3 & N_4 \end{bmatrix} \begin{Bmatrix} V_1 \\ \theta_1 \\ V_2 \\ \theta_2 \end{Bmatrix}$$

...…(2.5)

Where,

$$N_1 = 1 - \frac{3}{L^2}x^2 + \frac{2}{L^3}x^3;\; N_2 = x - \frac{2}{L}x^2 + \frac{x^3}{L^2};\; N_3 = \frac{3x^2}{L^2} - \frac{2x^3}{L^3} \text{ and } N_4 = -\frac{x^2}{L} + \frac{x^3}{L^2}$$

………(2.6)

N is called shape function which interpolates the beam displacement in terms of its nodal displacements.

1.2 Derivation of Element Stiffness Matrix

Now, the strain displacement relationship matrix [B] can be expressed from the following expressions with the help of eq (2.1)

$$\chi = \frac{d^2v}{dx^2} = \begin{bmatrix} 0 & 0 & 2 & 6x \end{bmatrix} \begin{Bmatrix} \alpha_0 \\ \alpha_1 \\ \alpha_2 \\ \alpha_4 \end{Bmatrix} = [B]\{\alpha\} = [B][A]^{-1}\{d\}$$

……..(2.2.1)

Where, $[B] = \begin{bmatrix} 0 & 0 & 2 & 6x \end{bmatrix}$; $[A] = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 1 & L & L^2 & L^3 \\ 0 & 1 & 2L & 3L^2 \end{bmatrix}$; $\{d\} = \begin{Bmatrix} V_1 \\ \theta_2 \\ V_2 \\ \theta_2 \end{Bmatrix}$

From the moment curvature relationship, we can write:

$$M = EI\chi = EI\frac{d^2v}{dx^2} = EI[B][A]^{-1}\{d\}$$

…….(2.2.2)

Strain energy,

$$U = \int_0^L \frac{1}{2}[\chi]^T[M]dx = \frac{EI}{2}\int_0^L \{d\}^T[A^{-1}]^T[B]^T[B][A^{-1}]\{d\}dx$$

……(2.2.3)

Thus,

$$\{F\} = \frac{\partial U}{\partial\{d\}} = EI\int_0^L [A^{-1}]^T[B]^T[B][A^{-1}]\{d\}dx$$

……..(2.2.4)

So, the stiffness matrix will be:

$$[k] = EI\int_0^L [A^{-1}]^T[B]^T[B][A^{-1}]dx = EI[A^{-1}]^T\int_0^L [B]^T[B]dx[A]^{-1}$$

…….(2.2.5)

Now, $\int_0^L [B^T][B]dx = \int_0^L \begin{Bmatrix} 0 \\ 0 \\ 2 \\ 6x \end{Bmatrix}[0 \quad 0 \quad 2 \quad 6x]dx = \int_0^L \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 4 & 12x \\ 0 & 0 & 12x & 36x^2 \end{bmatrix}dx = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 4L & 6L^2 \\ 0 & 0 & 6L^2 & 12L^3 \end{bmatrix}$

……….(2.2.6)

So,

$$[k]=EI\left[A^{-1}\right]^{T}\begin{bmatrix}0 & 0 & 0 & 0\\ 0 & 0 & 0 & 0\\ 0 & 0 & 4L & 6L^{2}\\ 0 & 0 & 6L^{2} & 12L^{3}\end{bmatrix}[A]^{-1}$$

$$[k]=EI\begin{bmatrix}1 & 0 & -\frac{3}{L^{2}} & \frac{2}{L^{3}}\\ 0 & 1 & -\frac{2}{L} & \frac{1}{L^{2}}\\ 0 & 0 & \frac{3}{L^{2}} & -\frac{2}{L^{3}}\\ 0 & 0 & -\frac{1}{L} & \frac{1}{L^{2}}\end{bmatrix}\begin{bmatrix}0 & 0 & 0 & 0\\ 0 & 0 & 0 & 0\\ 0 & 0 & 4L & 6L^{2}\\ 0 & 0 & 6L^{2} & 12L^{3}\end{bmatrix}\begin{bmatrix}1 & 0 & 0 & 0\\ 0 & 1 & 0 & 0\\ -\frac{3}{L^{2}} & -\frac{2}{L} & \frac{3}{L^{2}} & -\frac{1}{L}\\ \frac{2}{L^{3}} & \frac{1}{L^{2}} & -\frac{2}{L^{3}} & \frac{1}{L^{2}}\end{bmatrix}$$

$$=EI\begin{bmatrix}0 & 0 & 0 & 6\\ 0 & 0 & -2 & 0\\ 0 & 0 & 0 & -6\\ 0 & 0 & 2 & 6l\end{bmatrix}\begin{bmatrix}1 & 0 & 0 & 0\\ 0 & 1 & 0 & 0\\ -\frac{3}{L^{2}} & -\frac{2}{L} & \frac{3}{L^{2}} & -\frac{1}{L}\\ \frac{2}{L^{3}} & \frac{1}{L^{2}} & -\frac{2}{L^{3}} & \frac{1}{L^{2}}\end{bmatrix}=EI\begin{bmatrix}\frac{12}{L^{3}} & \frac{6}{L^{2}} & \frac{12}{L^{3}} & \frac{6}{L^{2}}\\ \frac{6}{L^{2}} & \frac{4}{L} & -\frac{6}{L^{2}} & \frac{2}{L}\\ -\frac{12}{L^{3}} & -\frac{6}{L^{2}} & \frac{12}{L^{3}} & -\frac{6}{L^{2}}\\ \frac{6}{L^{2}} & \frac{2}{L} & -\frac{6}{L^{2}} & \frac{4}{L}\end{bmatrix}$$

Thus, the element stiffness of a beam member is:

$$[k]=\frac{EI}{L^{3}}\begin{bmatrix}12 & 6L & -12 & 6L\\ 6L & 4L^{2} & -6L & 2L^{2}\\ -12 & -6L & 12 & -6L\\ 6L & 2L^{2} & -6L & 4L^{2}\end{bmatrix}$$

…...(2.2.7)

1.3 Equivalent Loading on Beam Member

In finite element analysis, the external loads are necessary to be acting at the joints, which does not happen always; as some forces may act on the member. The forces acting on the member should be replaced by equivalent forces acting at the joints. These joint forces obtained from the forces on the members are called equivalent joint loads. These joint loads are combined with the actual joint loads to provide the combined joint loads, which are then utilized in the analysis.

1.4 Varying Load

Let a beam is loaded with a linearly varying load as shown in the figure below. The equivalent forces at nodes can be expressed using finite element technique. If w(x) is the function of load, then the nodal load can be expressed as follows.

$$\{Q\} = \int [N]^T w(x)\,dx$$

………(2.4.1)

The loading function for the present case can be written as:

$$w(x) = w_1 + \frac{w_2 - w_1}{L} x$$

……..(2.4.2)

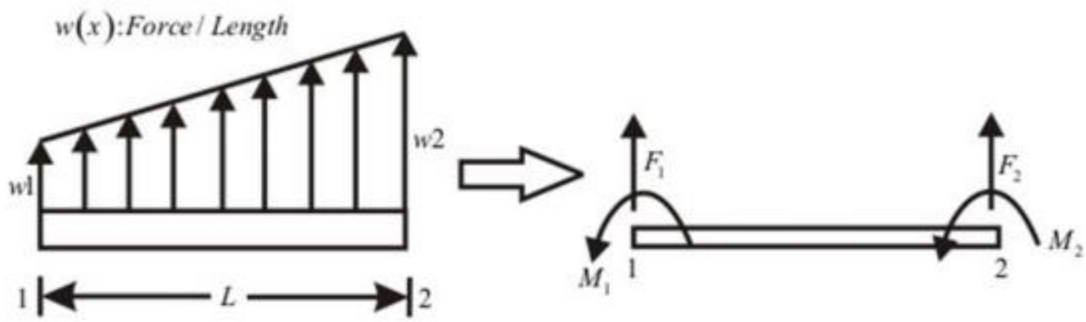


Fig. 2.4.1 Varying load on beam

From eqs. (2.4.1) and (2.4.2), the equivalent nodal load will become

$$\{Q\} = \begin{Bmatrix} F_1 \\ M_1 \\ F_2 \\ M_2 \end{Bmatrix} = \begin{Bmatrix} \int_0^L \left(\frac{2x^3}{L^3} - \frac{3x^2}{L^2} + 1 \right) w(x)dx \\ \int_0^L \left(\frac{x^3}{L^2} - \frac{2x^2}{L} + x \right) w(x)dx \\ \int_0^L \left(-\frac{2x^3}{L^3} + \frac{3x^2}{L^2} \right) w(x)dx \\ \int_0^L \left(\frac{x^3}{L^2} - \frac{x^2}{L} \right) w(x)dx \end{Bmatrix} = \begin{Bmatrix} \left(\frac{7w_1}{20} + \frac{3w_2}{20} \right) L \\ \left(\frac{w_1}{20} + \frac{w_2}{30} \right) L^2 \\ \left(\frac{3w_1}{20} + \frac{7w_2}{20} \right) L \\ \left(-\frac{w_1}{30} - \frac{w_2}{20} \right) L^2 \end{Bmatrix}$$

…………..(2.4.3)

Now, if $w_1 = w_2 = w$, then the equivalent nodal force will be:

$$\{Q\} = \begin{Bmatrix} \frac{wL}{2} \\ \frac{wL^2}{12} \\ \frac{wL}{2} \\ -\frac{wL^2}{12} \end{Bmatrix}$$

……...(2.4.4)

With the above approach, the equivalent nodal load can be found for various loading function acting on beam members.

1.5 Worked Out Example

Analyze the beam shown below by the finite element method. Assume the moment of inertia of member 2 as twice that of member 1. Find the bending moment and reactions at supports of the beam assuming the length of span, L as 4 m, concentrated load (P) as 15 kN and udl, w as 4 kN/m.

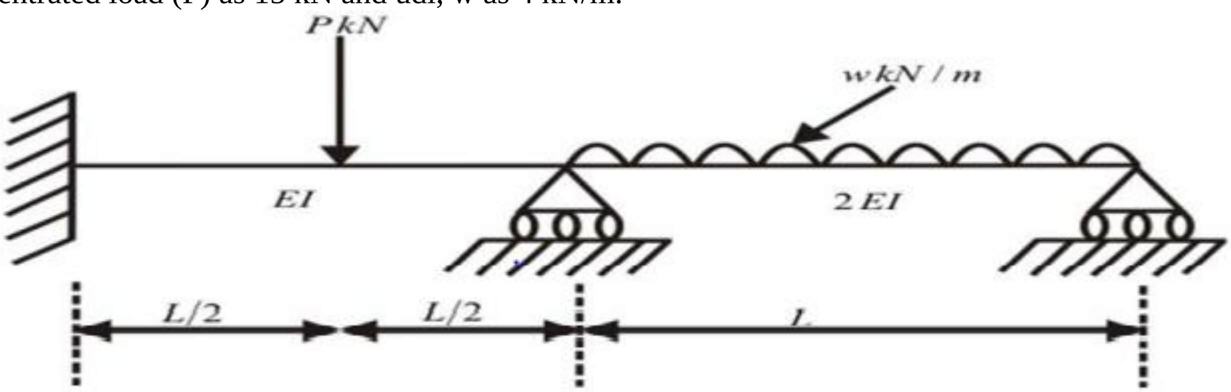


Fig 2.5.1 Example of a continuous beam

Solution

Step 1: Numbering of Nodes and Members The analysis of beam starts with the numbering of members and joints as shown below:

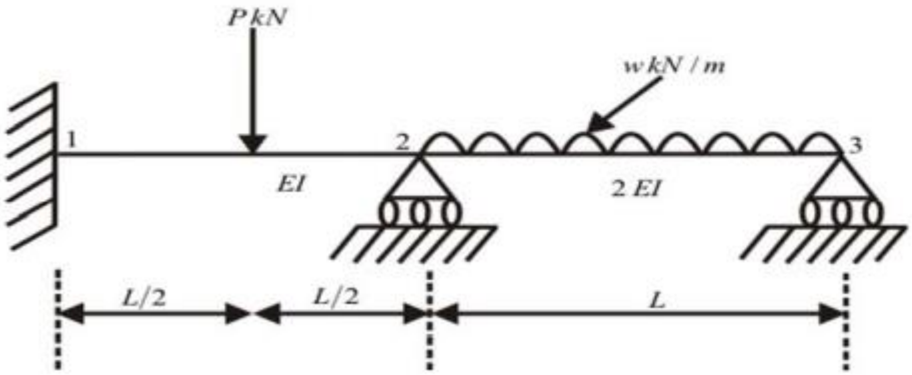


Fig 2.5.2 Numbering of nodes and members

The member AB and BC are designated as (1) and (2). The points A,B,C are designated by nodes 1, 2 and 4. The member information for beam is shown in tabulated form as shown in Table 4.4.1. The coordinate of node 1 is assumed as (0, 0). The coordinate and restraint joint information are shown in Table 4.4.2. The integer 1 in the restraint list indicates the restraint exists and 0 indicates the restraint at that particular direction does not exist. Thus, in node no. 2, the integer 0 in rotation indicates that the joint is free rotation.

Table 2.5.1 Member Information for Beam

Member number	Starting node	Ending node	Rigidity modulus
1	1	2	EI
2	2	3	2EI

Table 2.5.2 Nodal Information for Beam

Node No.	Coordinates		Restraint List	
	x	*y*	*Vertical*	*Rotation*
1	*0*	0	1	1
2	*L*	0	1	0
3	*2L*	0	1	0

Step 2: Formation of member stiffness matrix:

The local stiffness matrices of each member are given below based on their individual member properties and orientations. Thus the local stiffness matrix of member (1) is:

$$[k]_1 = \begin{array}{c} \\ \left[\begin{array}{cccc} \frac{12EI}{L^3} & \frac{6EI}{L^2} & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{4EI}{L} & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{2EI}{L} & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{array}\right] \end{array} \begin{array}{c} 1 \\ 2 \\ 3 \\ 4 \end{array}$$

(columns: 1, 2, 3, 4)

Similarly, the local stiffness matrix of member (2) is:

$$[k]_2 = \left[\begin{array}{cccc} \frac{24EI}{L^3} & \frac{12EI}{L^2} & -\frac{24EI}{L^3} & \frac{12EI}{L^2} \\ \frac{12EI}{L^2} & \frac{8EI}{L} & -\frac{12EI}{L^2} & \frac{4EI}{L} \\ -\frac{24EI}{L^3} & -\frac{12EI}{L^2} & \frac{24EI}{L^3} & -\frac{12EI}{L^2} \\ \frac{12EI}{L^2} & \frac{4EI}{L} & -\frac{12EI}{L^2} & \frac{8EI}{L} \end{array}\right] \begin{array}{c} 3 \\ 4 \\ 5 \\ 6 \end{array}$$

(columns: 3, 4, 5, 6)

Step 3: Formation of global stiffness matrix:

The global stiffness matrix is obtained by assembling the local stiffness matrix of members (1) and (2) as follows:

$$
\begin{array}{c}
\begin{matrix} \quad 1 & \quad 2 & \quad 3 & \quad 4 & \quad 5 & \quad 6 & \end{matrix} \\
[K] = \begin{bmatrix}
\frac{12EI}{L^3} & \frac{6EI}{L^2} & -\frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & 0 \\
\frac{6EI}{L^2} & \frac{4EI}{L} & -\frac{6EI}{L^2} & \frac{2EI}{L} & 0 & 0 \\
-\frac{12EI}{L^3} & -\frac{6EI}{L^2} & \frac{36EI}{L} & \frac{6EI}{L^2} & -\frac{24EI}{L^3} & \frac{12EI}{L^2} \\
\frac{6EI}{L^2} & \frac{2EI}{L} & \frac{6EI}{L^2} & \frac{12EI}{L} & -\frac{12EI}{L^2} & \frac{4EI}{L} \\
0 & 0 & -\frac{24EI}{L^3} & -\frac{12EI}{L^2} & \frac{24EI}{L^3} & -\frac{12EI}{L^2} \\
0 & 0 & \frac{12EI}{L^2} & \frac{4EI}{L} & -\frac{12EI}{L^2} & \frac{8EI}{L}
\end{bmatrix}
\begin{matrix} 1 \\ 2 \\ 3 \\ 4 \\ 5 \\ 6 \end{matrix}
\end{array}
$$

Step 4: Boundary condition:

The boundary conditions according to the support of the beam can be expressed in terms of the displacement vector. The displacement vector will be as follows

$$
\{d\} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ \theta_2 \\ 0 \\ \theta_3 \end{Bmatrix}
$$

Step 5: Load vector:

The concentrated load on member (1) and the distributed load on member (2) are replaced by equivalent joint load. The equivalent joint load vector can be written as

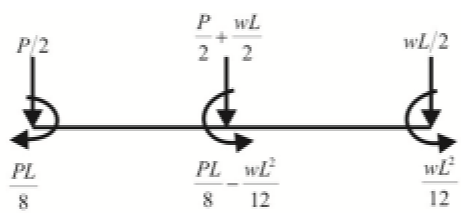


Fig. 2.5.3 Equivalent Load

$$\{F\} = \begin{Bmatrix} -\frac{P}{2} \\ -\frac{PL}{8} \\ -\left(\frac{P}{2}+\frac{wL}{2}\right) \\ \left(\frac{PL}{8}-\frac{wL^2}{12}\right) \\ -\frac{wL}{2} \\ \frac{wL^2}{12} \end{Bmatrix}$$

Step 6 : Determination of unknown displacements:.

The unknown displacement can be obtained from the relationship as given below:

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

$$\{d\} = [K]^{-1}\{F\}$$

$$\begin{Bmatrix} 0 \\ 0 \\ 0 \\ \theta_2 \\ 0 \\ \theta_3 \end{Bmatrix} = \begin{bmatrix} \frac{12EI}{L^3} & \frac{6EI}{L^2} & -\frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & 0 \\ \frac{6EI}{L^2} & \frac{4EI}{L} & -\frac{6EI}{L^2} & \frac{2EI}{L} & 0 & 0 \\ -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & \frac{36EI}{L} & \frac{6EI}{L^2} & -\frac{24EI}{L^3} & \frac{12EI}{L^2} \\ \frac{6EI}{L^2} & \frac{2EI}{L} & \frac{6EI}{L^2} & \frac{12EI}{L} & -\frac{12EI}{L^2} & \frac{4EI}{L} \\ 0 & 0 & -\frac{24EI}{L^3} & -\frac{12EI}{L^2} & \frac{24EI}{L^3} & -\frac{12EI}{L^2} \\ 0 & 0 & \frac{12EI}{L^2} & \frac{4EI}{L} & -\frac{12EI}{L^2} & \frac{8EI}{L} \end{bmatrix}^{-1} \times \begin{Bmatrix} -\frac{P}{2} \\ -\frac{PL}{8} \\ -\left(\frac{P}{2}+\frac{wL}{2}\right) \\ \left(\frac{PL}{8}-\frac{wL^2}{12}\right) \\ -\frac{wL}{2} \\ \frac{wL^2}{12} \end{Bmatrix}$$

The above relation may be condensed into following

$$\begin{Bmatrix} \theta_2 \\ \theta_3 \end{Bmatrix} = \begin{bmatrix} \frac{12EI}{L} & \frac{4EI}{L} \\ \frac{4EI}{L} & \frac{8EI}{L} \end{bmatrix}^{-1} \times \begin{Bmatrix} \frac{PL}{8}-\frac{wL^2}{12} \\ \frac{wL^2}{12} \end{Bmatrix} = \frac{L}{20EI}\begin{bmatrix} 2 & -1 \\ -1 & 3 \end{bmatrix}\begin{Bmatrix} \frac{PL}{8}-\frac{wL^2}{12} \\ \frac{wL^2}{12} \end{Bmatrix}$$

$$\begin{Bmatrix} \theta_2 \\ \theta_3 \end{Bmatrix} = \frac{L}{20\,EI}\begin{bmatrix} \frac{PL}{4}-\frac{wL^2}{4} \\ -\frac{PL}{8}+\frac{wL^2}{3} \end{bmatrix}$$

$$\theta_2 = \frac{PL^2}{80\,EI} - \frac{wL^3}{80\,EI}$$

$$\theta_3 = -\frac{PL}{160EI} + \frac{wL^3}{60EI}$$

Step 7: Determination of member end actions: The member end actions can be obtained from The corresponding member stiffness and the nodal displacements. The member end actions for each member are derived as shown below.

Member-(1)

$$\begin{Bmatrix} F_1 \\ M_1 \\ F_2 \\ M_2 \end{Bmatrix} = \frac{L}{20EI}\begin{bmatrix} \frac{12EI}{L^3} & \frac{6EI}{L^2} & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{4EI}{L} & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{2EI}{L} & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \times \begin{Bmatrix} 0 \\ 0 \\ 0 \\ \frac{PL}{4} - \frac{wL^2}{4} \end{Bmatrix} = \begin{Bmatrix} \frac{3P}{40} - \frac{3wL}{40} \\ \frac{PL}{40} - \frac{wL^2}{40} \\ -\frac{3P}{40} + \frac{3wL}{40} \\ \frac{PL}{20} - \frac{wL^2}{20} \end{Bmatrix}$$

Member-(2)

$$\begin{Bmatrix} F_2 \\ M_2 \\ F_3 \\ M_3 \end{Bmatrix} = \frac{EI}{L}\begin{bmatrix} \frac{24}{L^2} & \frac{12}{L} & -\frac{24}{L^2} & \frac{12}{L} \\ \frac{12}{L} & 8 & -\frac{12}{L} & 4 \\ -\frac{24}{L^2} & -\frac{12}{L} & \frac{24}{L^2} & -\frac{12}{L} \\ \frac{12}{L} & 4 & -\frac{12}{L} & 8 \end{bmatrix} \times \begin{Bmatrix} 0 \\ \frac{PL}{4} - \frac{wL^2}{4} \\ 0 \\ -\frac{PL}{8} + \frac{wL^2}{3} \end{Bmatrix} = \begin{Bmatrix} \frac{wL}{20} + \frac{3P}{40} \\ \frac{3PL}{40} - \frac{wL^2}{30} \\ -\frac{wL}{20} - \frac{3P}{40} \\ \frac{wL^2}{12} \end{Bmatrix}$$

Actual member end actions:

Member (1)

$$\begin{Bmatrix} \overline{F_1} \\ \overline{M_1} \\ \overline{F_2} \\ \overline{M_2} \end{Bmatrix} = \begin{Bmatrix} \frac{3P}{40} - \frac{3wL}{40} \\ \frac{PL}{40} - \frac{wL^2}{40} \\ -\frac{3P}{40} + \frac{3wL}{40} \\ \frac{PL}{20} - \frac{wL^2}{20} \end{Bmatrix} + \begin{Bmatrix} \frac{P}{2} \\ \frac{PL}{8} \\ \frac{P}{2} \\ -\frac{PL}{8} \end{Bmatrix} = \begin{Bmatrix} \frac{23P}{40} - \frac{3wL}{40} \\ \frac{6PL}{40} - \frac{wL^2}{40} \\ \frac{17P}{40} + \frac{3wL}{40} \\ -\frac{3PL}{40} - \frac{wL^2}{20} \end{Bmatrix}$$

Member (2)

$$\begin{Bmatrix} \overline{F_2} \\ \overline{M_2} \\ \overline{F_3} \\ \overline{M_3} \end{Bmatrix} = \begin{Bmatrix} \frac{wL}{20} + \frac{3P}{40} \\ \frac{3PL}{40} - \frac{wL^2}{30} \\ -\frac{wL}{20} - \frac{3P}{40} \\ \frac{wL^2}{12} \end{Bmatrix} + \begin{Bmatrix} \frac{wL}{2} \\ \frac{wL^2}{12} \\ \frac{wL}{2} \\ -\frac{wL^2}{12} \end{Bmatrix} = \begin{Bmatrix} \frac{11wL}{20} + \frac{3P}{40} \\ \frac{3PL}{40} + \frac{wL^2}{20} \\ \frac{9wL}{20} - \frac{3P}{40} \\ 0 \end{Bmatrix}$$

The support reactions at the supports A, B and C are $\{F_R\} = \begin{Bmatrix} R_A \\ R_B \\ R_C \end{Bmatrix} = \begin{Bmatrix} \frac{23P}{40} - \frac{3wL}{40} \\ \frac{25wL}{40} + \frac{P}{2} \\ \frac{9wL}{20} - \frac{3P}{40} \end{Bmatrix}$

Putting the numerical values of *L*, *P* and *w* (P=15, L=4, w=4) the member actions and support reactions will be as follows:

Member end actions:

$$\begin{Bmatrix} F_2 \\ M_2 \\ F_3 \\ M_3 \end{Bmatrix} = \begin{Bmatrix} 9.925 \\ 7.7 \\ 6.075 \\ 0 \end{Bmatrix} \quad , \quad \begin{Bmatrix} F_1 \\ M_1 \\ F_2 \\ M_2 \end{Bmatrix} = \begin{Bmatrix} 7.425 \\ 7.4 \\ 7.575 \\ -7.7 \end{Bmatrix}$$

Support reactions:

$$\{F_R\} = \begin{Bmatrix} R_A \\ R_B \\ R_C \end{Bmatrix} = \begin{Bmatrix} 7.425 \\ 17.5 \\ 6.075 \end{Bmatrix}$$

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

SCHOOL OF MECHANICAL ENGINEERING
DEPARTMENT OF AERONAUTICAL ENGINEERING

UNIT – III –FINITE ELEMENT METHOD FOR AIRCRAFT STRUCTURES – SAEA1504

UNIT – III
ANALYSIS OF TRUSSES AND FRAMES

Trusses Using FEA

We started this series of lectures looking at truss problems. We limited the discussion to statically determinate structures and solved for the forces in elements and reactions at supports using basic concepts from statics.

In this section, we will apply basic finite element techniques to solve general two dimensional truss problems. The technique is a little more complex than that originally used to solve truss problems, but it allows us to solve problems involving statically indeterminate structures.

3.1 Local and Global Coordinates

We start by looking at the beam or element shown in the diagram below. This element attaches to two nodes, 1 and 2. In the Figure we are showing two coordinate systems. One is a one dimensional coordinate system that aligns with the length of the element. We will call this the local coordinate system. The other is a two dimensional coordinate system that does not align with the element. We will call this the global coordinate system. The $\langle x', y' \rangle$ coordinates are the local coordinates for the element and $\langle x, y \rangle$ are the global coordinates.

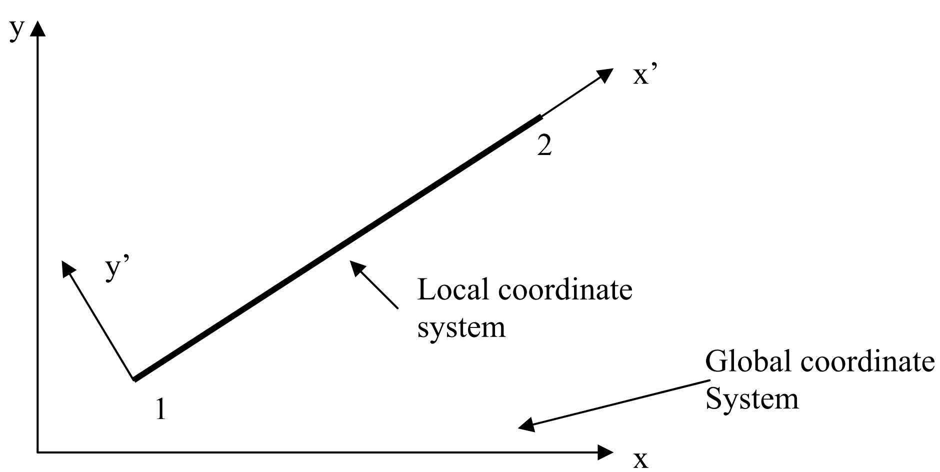


Figure 1 - Local and global coordinate systems

We can convert the displacements shown in the local coordinate system by looking at the following diagram. We will let q_1' and q_2' represent displacements in the local coordinate system and q_1, q_2, q_3, and q_4 represent displacements in the x-y (global) coordinate system. Note that the odd subscripted displacements are in the x direction and the even ones are in the y direction as shown in the following diagram.

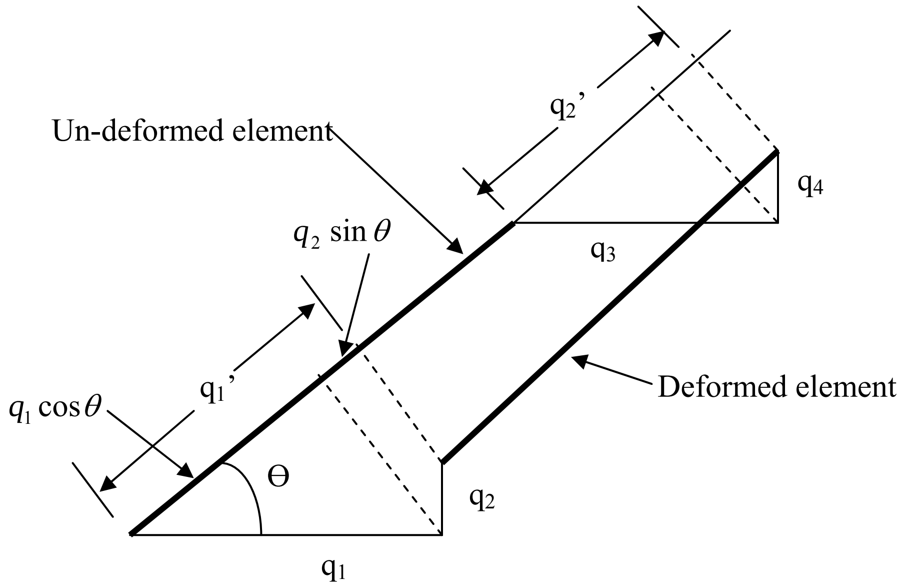


Figure 2 - The deformation of an element in both local and global coordinate systems.

We know that for small deformations in tension or compression a beam, acts like a spring. The amount of deformation is linearly proportional to the force applied to the beam. As the beam is stretched or compressed, we are added potential energy to the beam. This energy is called strain energy and it can be modeled with Hook's law. The law states that the force is directly proportional to the deformation.

$$F = k\Delta x \tag{3.1}$$

We can compute the energy by integrating over the deformation

$$u = k\int_0^Q x dx = \frac{1}{2}kQ^2 \tag{3.2}$$

where $k = \frac{AE}{L}$ the element stiffness, A = the cross sectional area of the element, E = Young's modulus for the material, and L = the length of the element. Q is the total change in length of the element. Note that we are assuming the deformation is linear over the element. All equal length segments of the element will deform the same amount. We call this a constant strain deformation of the element.

We can rewrite this change in length as

$$Q = (q_2^{'} - q_1^{'}) \tag{3.3}$$

Substituting this into equation (3.2) gives us

$$u = \frac{1}{2}k(q_2' - q_1')^2 \tag{3.4}$$

or expanding

$$u = \frac{1}{2} k(q_2'^2 - 2q_2'q_1' + q_1'^2) \tag{3.5}$$

Rewriting this in vector form we let

$$q' = \begin{Bmatrix} q_1' \\ q_2' \end{Bmatrix} \tag{3.6}$$

and

$$k' = \frac{AE}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{3.7}$$

With this we can rewrite equation (3.5) as:

$$u = \frac{1}{2} q'^T k' q' \tag{3.8}$$

We can do the indicated operations in (3.8) to see how the vector notation works. We do this by first expanding the terms then doing the multiplication.

$$u = \frac{AE}{2L} \{q_1' \quad q_2'\} \begin{Bmatrix} 1 & -1 \\ -1 & 1 \end{Bmatrix} \begin{Bmatrix} q_1' \\ q_2' \end{Bmatrix} \tag{3.9}$$

$$u = \frac{AE}{2L} \{q_1' - q_2' \quad -q_1' + q_2'\} \begin{Bmatrix} q_1' \\ q'_2 \end{Bmatrix} \tag{3.10}$$

$$u = \frac{AE}{2L} \left(q_1'(q_1' - q_2') + q_2'(q_2' - q_1') \right) \tag{3.11}$$

$$u = \frac{AE}{2L} \left(q_1'^2 - q_1'q_2' + q_2'^2 - q_1'q_2' \right) \tag{3.12}$$

$$u = \frac{AE}{2L} (q_1'^2 - 2q_1'q_2' + q_2'^2) \tag{3.13}$$

Which is the same as equation (3.5).

Equation (3.7) is the stiffness matrix for a one dimensional problem.

6.2 Two Dimensional Stiffness Matrix

We know for local coordinates that

$$q' = \begin{Bmatrix} q_1' \\ q_2' \end{Bmatrix} \tag{3.6}$$

and for global coordinates (See Figure 2)

$$q = \begin{Bmatrix} q_1 \\ q_2 \\ q_3 \\ q_4 \end{Bmatrix} \tag{3.14}$$

We can transform the global coordinates to local coordinates with the equations

$$q_1' = q_1 \cos\theta + q_2 \sin\theta \tag{3.15}$$

and

$$q_2' = q_3 \cos\theta + q_4 \sin\theta \tag{3.16}$$

This can be rewritten in vector notation as:

$$q' = Mq \tag{3.17}$$

where

$$M = \begin{bmatrix} c & s & 0 & 0 \\ 0 & 0 & c & s \end{bmatrix}, \tag{3.18}$$

$c = \cos\theta$, and $s = \sin\theta$.

Using

$$u = \frac{1}{2} q'^T k' q' \tag{3.8}$$

we can substitute in equation (3.17)

$$u = \frac{1}{2} q^T \left[M^T k' M \right] q \tag{3.19}$$

Now we will let

$$k = M^T k' M \qquad (3.20)$$

and doing the multiplication, k our stiffness matrix for global two dimensional coordinates becomes

$$k = \frac{AE}{L}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \qquad (3.21)$$

where:

E = Young's modulus for the element material
A = the cross sectional area of the element
L = the length of the element
$c = \cos\theta$
$s = \sin\theta$

3.3 Stress Computations

The stress can be written as

$$\sigma = E\varepsilon \qquad (3.22)$$

where ε is the strain, the change in length per unit of length. We can rewrite this as:

total deformation

$$\sigma = E\frac{q_2' - q_1'}{L} \qquad (3.23)$$

length of element

In vector form we can write the equation as

$$\sigma = \frac{E}{L}\{-1 \quad 1\}\begin{Bmatrix} q_1' \\ q_2' \end{Bmatrix} \qquad (3.24)$$

From our previous discussion, we know that in local coordinates

$$q' = \begin{Bmatrix} q_1' \\ q_2' \end{Bmatrix} \qquad (3.6)$$

and in global coordinates

$$q = \begin{Bmatrix} q_1 \\ q_2 \\ q_3 \\ q_4 \end{Bmatrix} \tag{3.14}$$

From equation (3.17) we know that

$$q' = Mq \tag{3.17}$$

where

$$M = \begin{bmatrix} c & s & 0 & 0 \\ 0 & 0 & c & s \end{bmatrix} \tag{3.18}$$

Substituting this in to the equation (3.24) yields

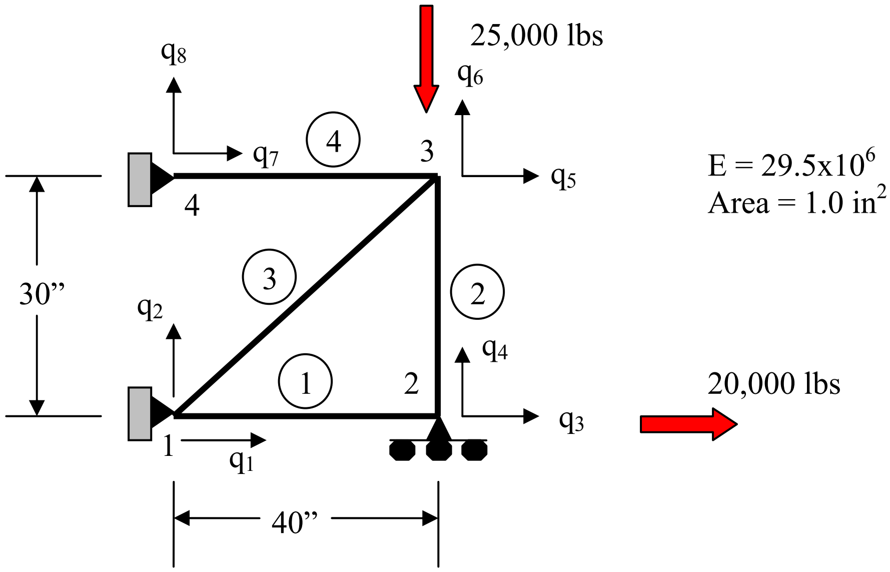
$$\sigma = \frac{E}{L}\{-1 \quad 1\}Mq \tag{3.25}$$

Now we multiply M by the vector

$$\sigma = \frac{E}{L}\{-c \quad -s \quad c \quad s\}q \tag{3.26}$$

3.4 Truss Example

We can now use the techniques we have developed to compute the stresses in a truss. Consider



Computing Displacements

There are 4 nodes and 4 elements making up the truss. We are going to do a two dimensional analysis so each node is constrained to move in only the X or Y direction. We call these directions of motion degrees of freedom or **dof** for short. There are 4 nodes and 8 degrees of freedom (two degrees of freedom for each node). We can number the degrees of freedom with the formulas:

Vertical degree of freedom $$dof = 2 * node \tag{3.27}$$

Horizontal degree of freedom $$dof = 2 * node - 1 \tag{3.28}$$

where *node* is the node number.

We can locate each node by its coordinates. The table below shows the coordinates of the nodes in the problem we are solving. We can use these coordinates to determine the lengths and angles of the elements.

Node	X	Y
1	0	0
2	40	0
3	40	30
4	0	30

Table 1 - Coordinates of the nodes in the truss.

Each element can be described as extending from one node to another. This also can be defined in a table below.

Element	From Node	To Node
1	1	2
2	3	2
3	1	3
4	4	3

Table 2 - The elements and the nodes they connect in the truss.

From these two tables we can derive the lengths of each element and the cosine and sine of their orientation. This is shown in the table below.

Element	Length	Cosine	Sine
1	40	1	0
2	30	0	-1
3	50	0.8	0.6
4	40	1	0

Table 3 - Elements with sines and cosines to be used in the stiffness matrix.

In the previous sections we developed the stiffness matrix for an element. This is shown in equation (3.21) below.

$$k = \frac{AE}{L}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \tag{3.21}$$

This stiffness matrix is for an element. The element attaches to two nodes and each of these nodes has two degrees of freedom. The rows and columns of the stiffness matrix correlate to those degrees of freedom.

Using the equation shown in (3.21) we can construct that stiffness matrix for element 1 defined in the table above. The stiffness matrix is:

Global dof

$$k_1 = \frac{29.5x10^6}{40}\begin{array}{c} \begin{matrix} 1 & 2 & 3 & 4 \end{matrix} \\ \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \end{array}\begin{matrix} 1 \\ 2 \\ 3 \\ 4 \end{matrix} \tag{3.29}$$

Element 2

$$k_2 = \frac{29.5x10^6}{30}\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix}\begin{matrix} 5 \\ 6 \\ 3 \\ 4 \end{matrix} \quad \text{columns: } 5\ 6\ 3\ 4 \tag{3.30}$$

Element 3

$$k_3 = \frac{29.5x10^6}{50}\begin{bmatrix} .64 & .48 & -.64 & -.48 \\ .48 & .36 & -.48 & -.36 \\ -.64 & -.48 & .64 & .48 \\ -.48 & -.36 & .48 & .36 \end{bmatrix}\begin{matrix} 1 \\ 2 \\ 5 \\ 6 \end{matrix} \quad \text{columns: } 1\ 2\ 5\ 6 \tag{3.31}$$

Element 4

$$k_4 = \frac{29.5x10^6}{40}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}\begin{matrix} 7 \\ 8 \\ 5 \\ 6 \end{matrix} \quad \text{columns: } 7\ 8\ 5\ 6 \tag{3.32}$$

The next step is to add the stiffness matrices for the elements to create a matrix for the entire structure. We can facilitate this by creating a common factor for Young's modulus and the length of the elements.

For element 1, we divide the outside by 15 and multiply each element of the matrix by 15. Multiplying and dividing by the same number is the same as multiplying and dividing by 1.

$$k_1 = \frac{29.5x10^6}{600}\begin{bmatrix} 15 & 0 & -15 & 0 \\ 0 & 0 & 0 & 0 \\ -15 & 0 & 15 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}\begin{matrix} 1 \\ 2 \\ 3 \\ 4 \end{matrix} \quad \text{columns: } 1\ 2\ 3\ 4 \tag{3.33}$$

We multiply and divide element 2 by 20.

$$k_2 = \frac{29.5x10^6}{600}\begin{array}{c}\begin{matrix}5 & 6 & 3 & 4\end{matrix}\\ \begin{bmatrix}0 & 0 & 0 & 0\\ 0 & 20 & 0 & -20\\ 0 & 0 & 0 & 0\\ 0 & -20 & 0 & 20\end{bmatrix}\end{array}\begin{matrix}5\\6\\3\\4\end{matrix} \qquad (3.34)$$

Multiply and divide element 3 by 12.

$$k_3 = \frac{29.5x10^6}{600}\begin{array}{c}\begin{matrix}1 & 2 & 5 & 6\end{matrix}\\ \begin{bmatrix}7.68 & 5.76 & -7.68 & -5.76\\ 5.76 & 4.32 & -5.76 & -4.32\\ 7.68 & 5.76 & 7.68 & 5.76\\ -5.76 & -4.32 & 5.76 & 4.32\end{bmatrix}\end{array}\begin{matrix}1\\2\\5\\6\end{matrix} \qquad (3.35)$$

We do the same for element 4 by multiplying and dividing it by 15.

$$k_4 = \frac{29.5x10^6}{600}\begin{array}{c}\begin{matrix}7 & 8 & 5 & 6\end{matrix}\\ \begin{bmatrix}15 & 0 & -15 & 0\\ 0 & 0 & 0 & 0\\ -15 & 0 & 15 & 0\\ 0 & 0 & 0 & 0\end{bmatrix}\end{array}\begin{matrix}7\\8\\5\\6\end{matrix} \qquad (3.36)$$

The coefficient for each stiffness matrix is the same so we can easily add the matrices. We add the degree of freedom for each element stiffness matrix into the same degree of freedom in the structural matrix. The resulting structural stiffness matrix is shown below.

$$K = \frac{29.5\times10^6}{600}\begin{array}{c}\begin{matrix}1 & 2 & 3 & 4 & 5 & 6 & 7 & 8\end{matrix}\\ \begin{bmatrix}22.68 & 5.76 & -15.0 & 0 & -7.68 & -5.76 & 0 & 0\\ 5.76 & 4.32 & 0 & 0 & -5.76 & -4.32 & 0 & 0\\ -15.0 & 0 & 15.0 & 0 & 0 & 0 & 0 & 0\\ 0 & 0 & 0 & 20.0 & 0 & -20.0 & 0 & 0\\ -7.68 & -5.76 & 0 & 0 & 22.68 & 5.76 & -15 & 0\\ -5.76 & 4.32 & 0 & -20.0 & 5.76 & 24.32 & 0 & 0\\ 0 & 0 & 0 & 0 & -15.0 & 0 & 15.0 & 0\\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0\end{bmatrix}\end{array}\begin{matrix}1\\2\\3\\4\\5\\6\\7\\8\end{matrix} \qquad (3.37)$$

Remembering our basic equation

$$KQ = F \tag{3.38}$$

where K is the structural or global stiffness matrix, Q is the displacement of each node, and F is the external force matrix. This results in

$$\frac{29.5\times10^{6}}{600}\begin{bmatrix} 22.68 & 5.76 & -15.0 & 0 & -7.68 & -5.76 & 0 & 0 \\ 5.76 & 4.32 & 0 & 0 & -5.76 & -4.32 & 0 & 0 \\ -15.0 & 0 & 15.0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 20.0 & 0 & -20.0 & 0 & 0 \\ -7.68 & -5.76 & 0 & 0 & 22.68 & 5.76 & -15 & 0 \\ -5.76 & 4.32 & 0 & -20.0 & 5.76 & 24.32 & 0 & 0 \\ 0 & 0 & 0 & 0 & -15.0 & 0 & 15.0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}\begin{Bmatrix} q_1 \\ q_2 \\ q_3 \\ q_4 \\ q_5 \\ q_6 \\ q_7 \\ q_8 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 20{,}000 \\ 0 \\ 0 \\ -25{,}000 \\ 0 \\ 0 \end{Bmatrix} \tag{3.39}$$

We have boundary conditions at the fixed supports. Our assumption is that these joints will not move in the constrained direction. We remove these from our matrix. The constrained displacements are dof 1, 2, 4, 7, and 8. The lines in equation (3.40) show the rows and columns that are removed.

$$\frac{29.5\times10^{6}}{600}\begin{bmatrix} 22.68 & 5.76 & -15.0 & 0 & -7.68 & -5.76 & 0 & 0 \\ 5.76 & 4.32 & 0 & 0 & -5.76 & -4.32 & 0 & 0 \\ -15.0 & 0 & 15.0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 20.0 & 0 & -20.0 & 0 & 0 \\ -7.68 & -5.76 & 0 & 0 & 22.68 & 5.76 & -15 & 0 \\ -5.76 & 4.32 & 0 & -20.0 & 5.76 & 24.32 & 0 & 0 \\ 0 & 0 & 0 & 0 & -15.0 & 0 & 15.0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}\begin{Bmatrix} q_1 \\ q_2 \\ q_3 \\ q_4 \\ q_5 \\ q_6 \\ q_7 \\ q_8 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 20{,}000 \\ 0 \\ 0 \\ -25{,}000 \\ 0 \\ 0 \end{Bmatrix} \tag{3.40}$$

The resulting matrix is:

$$\frac{29.5\times10^{6}}{600}\begin{bmatrix} 15 & 0 & 0 \\ 0 & 22.68 & 5.76 \\ 0 & 5.76 & 24.32 \end{bmatrix}\begin{Bmatrix} q_3 \\ q_5 \\ q_6 \end{Bmatrix} = \begin{Bmatrix} 20{,}000 \\ 0 \\ -25{,}000 \end{Bmatrix} \tag{3.41}$$

We can use Gaussian elimination or any number of other solution techniques to solve the system of equations shown above. Doing so yields

$$\begin{Bmatrix} q_3 \\ q_5 \\ q_6 \end{Bmatrix} = \begin{Bmatrix} 27.12\times10^{-3} \\ 5.65\times10^{-3} \\ -22.25\times10^{-3} \end{Bmatrix} \quad \text{inches} \tag{3.42}$$

Computing Stresses

Previously we showed that

$$\sigma = \frac{E}{L}\{-c \quad -s \quad c \quad s\}q \tag{3.26}$$

We use this equation to compute the stress in each element.

$$\sigma_1 = \frac{29.5\times10^6}{40}\{-1 \quad 0 \quad 1 \quad 0\}\begin{Bmatrix} 0 \\ 0 \\ 27.12\times10^{-3} \\ 0 \end{Bmatrix}\begin{matrix} 1 \\ 2 \\ 3 \\ 4 \end{matrix} \leftarrow \text{dof} \tag{3.43}$$

or

$$\sigma_1 = 20{,}000\,psi \tag{3.44}$$

$$\sigma_2 = \frac{29.5\times10^6}{30}\{0 \quad 1 \quad 0 \quad -1\}\begin{Bmatrix} 5.65\times10^{-3} \\ -22.25\times10^{-3} \\ -27.12\times10^{-3} \\ 0 \end{Bmatrix}\begin{matrix} 5 \\ 6 \\ 3 \\ 4 \end{matrix} \tag{3.45}$$

$$\sigma_2 = -21{,}875\,psi \tag{3.46}$$

Using a similar technique we get

$$\sigma_3 = -5{,}208\,psi \tag{3.47}$$

and

$$\sigma_4 = 4{,}167\,psi \tag{3.48}$$

Computing the Reactions

The last step is to compute the support reactions. We need to determine the reaction forces along dof 1, 2, 3, 7, and 8 which correspond to the fixed supports. These are obtained by substituting Q into the original finite element equation.

$$R = KQ - F \tag{3.48}$$

We only need to use those rows of the structural stiffness matrix that correspond to the fixed supports. At these supports, we are not supplying an external force so F=0. Our equation becomes

$$R = KQ \tag{3.50}$$

or

$$\begin{Bmatrix} R_1 \\ R_2 \\ R_4 \\ R_7 \\ R_8 \end{Bmatrix} = \frac{29.5\times10^6}{600}\begin{bmatrix} 22.68 & 5.76 & -15.0 & 0 & -7.68 & -5.76 & 0 & 0 \\ 5.76 & 4.32 & 0 & 0 & -5.76 & -4.32 & 0 & 0 \\ 0 & 0 & 0 & 20 & 0 & -20 & 0 & 0 \\ 0 & 0 & 0 & 0 & -15.0 & 0 & 15.0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}\begin{Bmatrix} 0 \\ 0 \\ 27.12\times10^{-3} \\ 0 \\ 5.65\times10^{-3} \\ -22.25\times10^{-3} \\ 0 \\ 0 \end{Bmatrix} \tag{3.51}$$

We multiply the stiffness matrix K and the deformation vector Q to get the reactions. They are shown in the following equation.

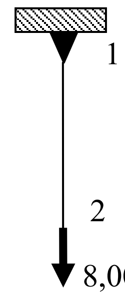
$$\begin{Bmatrix} R_1 \\ R_2 \\ R_4 \\ R_7 \\ R_8 \end{Bmatrix} = \begin{Bmatrix} -15{,}833.3 \\ 3{,}126 \\ 21{,}879 \\ -4{,}167 \\ 0 \end{Bmatrix} \tag{3.52}$$

Problems

1. Element area = 1.5 in^2 E=30,000,000

Element length = 5 feet

Write the stiffness matrix for the structure. The bar is vertical. Show all work.



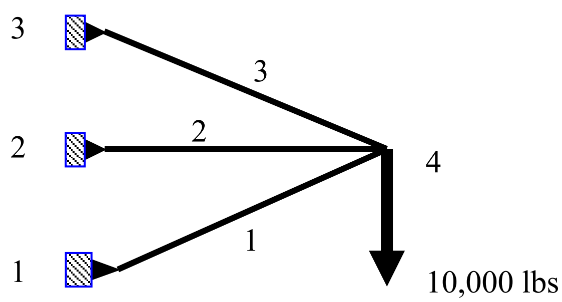
2. Using a different load, the element shown in Problem 1 deforms by 0.02 inches in length. What is the stress in the material? Use a finite element approach to solve the problem. Show all work.

3. Use a finite element approach, solve for the stress, joint displacement, and reaction force on the element shown in Problem 1. Use the 8,000 lbs force as shown in the diagram. Show all work.

4. The structure shown in the diagram results in the stiffness matrix shown in the table. Manually solve for the displacement of node 4. Show all work.

1.0e+006 *

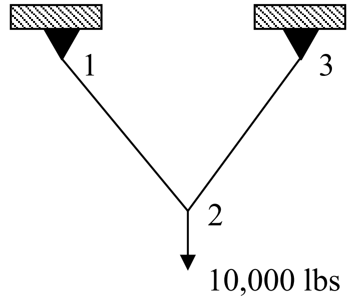
0.6293	0.4720	0	0	0	0	-0.6293	-0.4720
0.4720	0.3540	0	0	0	0	-0.4720	-0.3540
0	0	0.6146	0	0	0	-0.6146	0
0	0	0	0	0	0	0	0
0	0	0	0	0.6293	-0.4720	-0.6293	0.4720
0	0	0	0	-0.4720	0.3540	0.4720	-0.3540
-0.6293	-0.4720	-0.6146	0	-0.6293	0.4720	1.8733	0
-0.4720	-0.3540	0	0	0.4720	-0.3540	0	0.7080



Element	**Area**	**E**
1	2 in^2	29.5e6
2	1 in^2	29.5e6
3	2 in^2	29.5e6
Node	**X feet**	**Y feet**
1	0	0
2	0	3
3	0	6
4	4	3

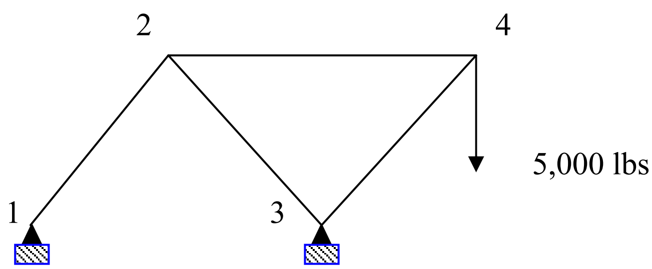
5. Element area = 1 in^2 Material = steel

Node	**X**	**Y**
1	0	40
2	30	0
3	60	40



A. Find the joint displacements
B. Find the stress in the elements
C. Find the reactions

6. Element area = 1 in^2 Material = steel



Node	**X**	**Y**
1	0	0
2	4	3
3	8	0
4	12	3
Element	**From Node**	**To Node**
1	1	2
2	2	3
3	2	4
4	3	4

D. Find the joint displacements
E. Find the stress in the elements
F. Find the reactions

Write a Matlab program that uses the finite element technique discussed in class to solve for the displacements, stresses, and reactions in a finite element truss. You may want to modify the static stress program you wrote earlier to create this new program. The two programs should be able to use the same input file.

Solve the problem shown above to turn in. Use both this new program and the static truss program to run the data file. Compare the results.

3.5.1 Plane Frame Analysis

The plane frame is a combination of plane truss and two dimensional beam. All the members lie in the same plane and are interconnected by rigid joints in case of plane frame. The internal stress resultants at a cross-section of a plane frame member consist of axial force, bending moment and shear force.

3.5.2 Member Stiffness Matrix

In case of plane frame, the degrees of freedom at each node will be (i) axial deformation, (ii) vertical deformation and (iii) rotation. Thus the frame members have three degrees of freedom at each node as shown in Fig. 4.5.1 below.

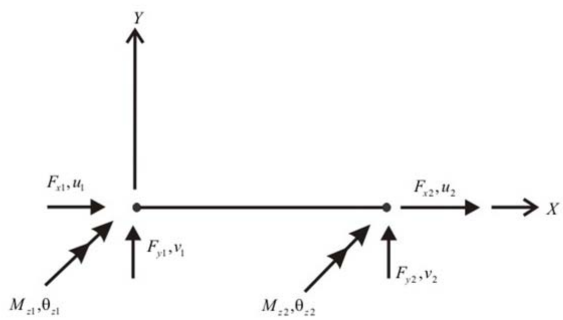


Fig. 4.5.1 Plane frame element

Therefore, the stiffness matrix of the frame in its local coordinate system will be the combination of 2-d truss and 2-d beam matrices:

$$\begin{matrix} u_1 & v_1 & \theta_1 & u_2 & v_2 & \theta_2 \end{matrix}$$

$$\overline{[k]}=\begin{bmatrix} \frac{AE}{L} & 0 & 0 & -\frac{AE}{L} & 0 & 0 \\ 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{AE}{L} & 0 & 0 & \frac{AE}{L} & 0 & 0 \\ 0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \quad (3.5.1)$$

3.5.3 Generalized Stiffness Matrix

In plane frame the members are oriented in different directions and hence it is necessary to transform stiffness matrix of individual members from local to global co-ordinate system before formulating the global stiffness matrix by assembly. The generalized stiffness matrix of a frame member can be obtained by transferring the matrix of local coordinate system into its global coordinate system. The transformation matrix can be expressed as:

$$[T]=\begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 & 0 & 0 \\ -\sin\theta & \cos\theta & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos\theta & \sin\theta & 0 \\ 0 & 0 & 0 & -\sin\theta & \cos\theta & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \tag{3.5.2}$$

Now, the generalized stiffness matrix of the member can be obtained from the relation of $[K] = [T]^T[\bar{K}][T]$. Thus considering $\lambda = \cos\theta$ and $\mu = \sin\theta$ the stiffness matrix in global coordinate system can be written as follows:

$$[K] = EI \begin{bmatrix} \lambda & -\mu & 0 & 0 & 0 & 0 \\ \mu & \lambda & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \lambda & -\mu & 0 \\ 0 & 0 & 0 & \mu & \lambda & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \times \begin{bmatrix} \frac{AE}{L} & 0 & 0 & -\frac{AE}{L} & 0 & 0 \\ 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{AE}{L} & 0 & 0 & \frac{AE}{L} & 0 & 0 \\ 0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix}$$

$$\times \begin{bmatrix} \lambda & \mu & 0 & 0 & 0 & 0 \\ -\mu & \lambda & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \lambda & \mu & 0 \\ 0 & 0 & 0 & -\mu & \lambda & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

$$= \begin{bmatrix} \left(\frac{EA}{L}\lambda^2 + \frac{12EI}{L^3}\mu^2\right) & \left(\frac{EA}{L}\lambda\mu - \frac{12EI}{L^3}\lambda\mu\right) & -\frac{6EI}{L^2}\mu & \left(-\frac{EA}{L}\lambda^2 - \frac{12EI}{L^3}\mu^2\right) & \left(-\frac{EA}{L}\lambda\mu + \frac{12EI}{L^3}\lambda\mu\right) & -\frac{6EI}{L^2}\mu \\ \left(\frac{EA}{L}\lambda\mu - \frac{12EI}{L^3}\lambda\mu\right) & \left(\frac{EA}{L}\mu^2 + \frac{12EI}{L^3}\lambda^2\right) & \frac{6EI}{L^2}\lambda & \left(-\frac{EA}{L}\lambda\mu + \frac{12EI}{L^3}\lambda\mu\right) & \left(-\frac{EA}{L}\mu^2 - \frac{12EI}{L^3}\lambda^2\right) & \frac{6EI}{L^2}\lambda \\ -\frac{6EI}{L^2}\mu & \frac{6EI}{L^2}\lambda & \frac{4EI}{L} & \frac{6EI}{L^2}\mu & -\frac{6EI}{L^2}\lambda & \frac{2EI}{L} \\ \left(-\frac{EA}{L}\lambda^2 - \frac{12EI}{L^3}\mu^2\right) & \left(-\frac{EA}{L}\lambda\mu + \frac{12EI}{L^3}\lambda\mu\right) & \frac{6EI}{L^2}\mu & \left(\frac{EA}{L}\lambda^2 + \frac{12EI}{L^3}\mu^2\right) & \left(\frac{EA}{L}\lambda\mu - \frac{12EI}{L^3}\lambda\mu\right) & \frac{6EI}{L^2}\mu \\ \left(-\frac{EA}{L}\lambda\mu + \frac{12EI}{L^3}\lambda\mu\right) & \left(-\frac{EA}{L}\mu^2 - \frac{12EI}{L^3}\lambda^2\right) & -\frac{6EI}{L^2}\lambda & \left(\frac{EA}{L}\lambda\mu - \frac{12EI}{L^3}\lambda\mu\right) & \left(\frac{EA}{L}\mu^2 + \frac{12EI}{L^3}\lambda^2\right) & -\frac{6EI}{L^2}\lambda \\ -\frac{6EI}{L^2}\mu & \frac{6EI}{L^2}\lambda & \frac{2EI}{L} & \frac{6EI}{L^2}\mu & -\frac{6EI}{L^2}\lambda & \frac{4EI}{L} \end{bmatrix}$$

(3.5.3)

3.5.4 Worked Out Example

Analyse the plane frame shown below. Assume the modulus of elasticity of the horizontal member is 1.5 times that of the vertical member and length of the vertical member is 1.5 times that of horizontal member. Find the bending moment and reactions at support assuming the length, cross section area and modulus of elasticity of vertical member as 3.0 m, 0.4 x 0.4 m^2 and 2 x 10^{11} N/mm^2, respectively.

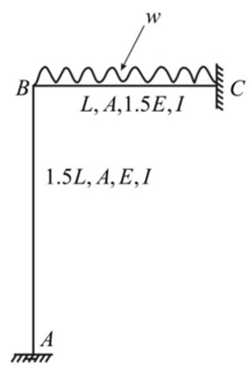


Fig. 3.5.2 Plane frame

<u>Solution</u>

Step 1: Numbering of Nodes and Members

The numbering of members and joints of the plane frame are as shown below:

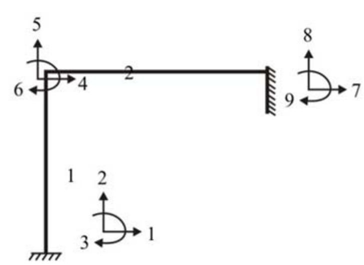


Fig. 3.5.3 Numbering of Nodes and Members

The members **AB** and **BC** are designated as (**1**) and (**2**). The points **A, B** and **C** are designated by nodes 1, 2 and 3. The member information for the frame is shown in tabulated form as shown in Table 1(a). The coordinate of node 1 is assumed as (0,0). The coordinate and restraint joint information are shown in Table 1(b). The integer 1 in the restraint list indicates the restraint exists and 0 indicates the restraint at that particular direction does not exist. Thus, in node no. 2, the integer 0 all the restraint type indicates that the joint is in free all the three directions.

Table 3.5.1 Member Information for Beam

Member number	Starting node	Ending node	Rigidity modulus
1	1	2	EI
2	2	3	1.5EI

Table 3.5.2 Nodal Information for Beam

Node no.	Coordinates		Restraint list		
	X	Y	Axial	Vertical	Rotation
1	0	0	1	1	1
2	0	1.5L	0	0	0
3	L	1.5L	1	1	1

Step 2: Formation of member stiffness matrix:

The individual member stiffness matrices can be found out directly from eqn. shown above. Thus the stiffness matrices of each member in global coordinate system are given below based on their individual member properties and orientations. Thus the stiffness matrix of member (1) is:

$$\begin{matrix} & 1 & 2 & 3 & 4 & 5 & 6 & \end{matrix}$$

$$[k]_1=\begin{bmatrix} \frac{12EI}{(1.5L)^3} & 0 & -\frac{6EI}{(1.5L)^2} & -\frac{12EI}{(1.5L)^3} & 0 & -\frac{6EI}{(1.5L)^2} \\ 0 & \frac{AE}{(1.5L)} & 0 & 0 & -\frac{AE}{(1.5L)} & 0 \\ -\frac{6EI}{(1.5L)^2} & 0 & \frac{4EI}{(1.5L)} & \frac{6EI}{(1.5L)^2} & 0 & \frac{2EI}{1.5L} \\ -\frac{12EI}{(1.5L)^3} & 0 & \frac{6EI}{(1.5L)^2} & \frac{12EI}{(1.5L)^3} & 0 & \frac{6EI}{(1.5L)^2} \\ 0 & -\frac{AE}{(1.5L)} & 0 & 0 & \frac{AE}{(1.5L)} & 0 \\ -\frac{6EI}{(1.5L)^2} & 0 & \frac{2EI}{(1.5L)} & \frac{6EI}{(1.5L)^2} & 0 & \frac{4EI}{(1.5L)} \end{bmatrix}\begin{matrix} 1 \\ 2 \\ 3 \\ 4 \\ 5 \\ 6 \end{matrix}$$

Similarly, the stiffness matrix of member (2) is :

$$\begin{matrix} 4 & 5 & 6 & 7 & 8 & 9 \end{matrix}$$

$$[k]_2 = \begin{bmatrix} \frac{A(1.5\,E)}{L} & 0 & 0 & -\frac{A(1.5\,E)}{L} & 0 & 0 \\ 0 & \frac{12(1.5\,E)I}{L^3} & \frac{6(1.5\,E)I}{L^2} & 0 & -\frac{12(1.5\,E)I}{L^3} & \frac{6(1.5\,E)I}{L^2} \\ 0 & \frac{6(1.5\,E)I}{L^2} & \frac{4(1.5\,E)I}{L} & 0 & -\frac{6(1.5\,E)I}{L^2} & \frac{2(1.5\,E)I}{L} \\ -\frac{A(1.5\,E)}{L} & 0 & 0 & \frac{A(1.5\,E)}{L} & 0 & 0 \\ 0 & -\frac{12(1.5\,E)I}{L^3} & -\frac{6(1.5\,E)I}{L^2} & 0 & \frac{12(1.5\,E)I}{L^3} & -\frac{6(1.5\,E)I}{L^2} \\ 0 & \frac{6(1.5\,E)I}{L^2} & \frac{2(1.5\,E)I}{L} & 0 & -\frac{6(1.5\,E)I}{L^2} & \frac{4(1.5\,E)I}{L} \end{bmatrix} \begin{matrix} 4 \\ 5 \\ 6 \\ 7 \\ 8 \\ 9 \end{matrix}$$

Step 3 : Formulation of global stiffness matrix:

The global stiffness matrix is obtained by assembling by assembling the local stiffness matrix of member (1) and (2) as follows:

$$\begin{matrix} 1 & 2 & 3 & 4 & 5 & 6 & 7 & 8 & 9 \end{matrix}$$

$$[K] = \begin{bmatrix} \frac{32EI}{9L^3} & 0 & -\frac{8EI}{3L^2} & -\frac{32EI}{9L^3} & 0 & -\frac{8EI}{3L^2} & 0 & 0 & 0 \\ 0 & \frac{2AE}{3L} & 0 & 0 & -\frac{2AE}{3L} & 0 & 0 & 0 & 0 \\ \frac{8EI}{3L^2} & 0 & \frac{8EI}{3L} & \frac{8EI}{3L^2} & 0 & \frac{4EI}{3L} & 0 & 0 & 0 \\ -\frac{32EI}{9L^3} & 0 & \frac{8EI}{3L^2} & (\frac{32EI}{9L^3} + \frac{1.5EA}{L}) & 0 & \frac{8EI}{3L^2} & -\frac{1.5EA}{L} & 0 & 0 \\ 0 & -\frac{2AE}{3L} & 0 & 0 & (\frac{2AE}{3L} + \frac{18EI}{L^3}) & \frac{9EI}{L^2} & 0 & -\frac{18EI}{L^3} & \frac{9EI}{L^2} \\ -\frac{8EI}{3L^2} & 0 & \frac{4EI}{3L} & \frac{8EI}{3L^2} & \frac{9EI}{L^2} & (\frac{8EI}{3L^2} + \frac{6EI}{L}) & 0 & -\frac{9EI}{L^2} & \frac{3EI}{L} \\ 0 & 0 & 0 & -\frac{1.5AE}{L} & 0 & 0 & \frac{1.5AE}{L} & 0 & 0 \\ 0 & 0 & 0 & 0 & -\frac{18EI}{L^3} & -\frac{9EI}{L^2} & 0 & \frac{18EI}{L^3} & -\frac{9EI}{L^2} \\ 0 & 0 & 0 & 0 & \frac{9EI}{L^2} & \frac{3EI}{L} & 0 & -\frac{9EI}{L^2} & \frac{6EI}{L} \end{bmatrix} \begin{matrix} 1 \\ 2 \\ 3 \\ 4 \\ 5 \\ 6 \\ 7 \\ 8 \\ 9 \end{matrix}$$

Step 4: Boundary conditions:

The boundary conditions according to the support of the frame can be expressed in terms of the displacement vector. The displacement vector will be as follows:

$$\{d\} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ \delta x_B \\ \delta y_B \\ \theta_B \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

Here, δx_B , δy_B and θ_B indicate the displacement in X-direction, displacement in Y-direction and rotation at point B.

Step 5: Load vector:

The distributed load on member (2) can be replaced by its equivalent joint load as shown in the figure below.

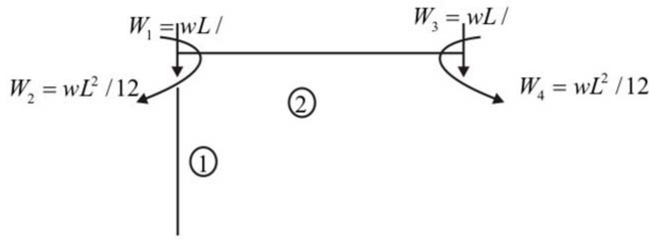


Fig. 3.5.4 Equivalent Joint Loads

Thus, the equivalent joint load vector can be written as

$$\{F\} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ -\dfrac{wL}{2} \\ -\dfrac{wL^2}{12} \\ 0 \\ -\dfrac{wL}{2} \\ \dfrac{wL^2}{12} \end{bmatrix}$$

Step 6: **Determination of unknown displacements:**

The unknown displacements can be obtained from the relationship of {F} = [K]{d} or {d} =[k]$^{-1}$ {F}. Now eliminating the rows and columns in the stiffness matrix and force matrix, corresponding to zero elements in displacement matrix, the reduced matrix will be as follows.

$$\begin{bmatrix} \delta x_B \\ \delta y_B \\ \theta_B \end{bmatrix} = \begin{bmatrix} \left(\dfrac{32EI}{9L^3} + \dfrac{1.5EA}{L}\right) & 0 & \dfrac{8EI}{3L^2} \\ 0 & \left(\dfrac{2AE}{3L} + \dfrac{18EI}{L^3}\right) & \dfrac{9EI}{L^2} \\ \dfrac{8EI}{3L^2} & \dfrac{9EI}{L^2} & \left(\dfrac{8EI}{3L} + \dfrac{6EI}{L}\right) \end{bmatrix}^{-1} \begin{bmatrix} 0 \\ -\dfrac{wL}{2} \\ -\dfrac{wL^2}{12} \end{bmatrix}$$

Thus, the unknown displacements will be:

$$\begin{bmatrix} \delta x_B \\ \delta y_B \\ \theta_B \end{bmatrix} = \frac{1}{10^{10}} \begin{bmatrix} 0.04327w \\ -1.7127w \\ -5.4978w \end{bmatrix}$$

Step 7: Determination of member end actions:

The member end actions can be obtained from the corresponding member stiffness and the nodal displacements. The member end actions for each member are derived as shown below.

Member – (1)

In case of member (1), the member forces will be: $\{F_m\}_1 = [K]_{(1)}\{d\}_{(1)}$

$$\begin{bmatrix} F_{x_1} \\ F_{y_1} \\ M_1 \\ F_{x_2} \\ F_{y_2} \\ M_2 \end{bmatrix} = 10^6 \begin{bmatrix} 56.17 & 0 & -126.4 & -56.17 & 0 & -126.4 \\ 0 & 7110 & 0 & 0 & -7110 & 0 \\ -126.4 & 0 & 379.2 & 126.4 & 0 & 189.6 \\ -56.17 & 0 & 379.2 & 56.17 & 0 & 126.4 \\ 0 & -7110 & 0 & 0 & 7110 & 0 \\ -126.4 & 0 & 189.6 & 126.4 & 0 & 379.2 \end{bmatrix} \times \begin{bmatrix} 0 \\ 0 \\ 0 \\ 4.327\,X\,10^{-12}w \\ -1.7127\,X\,10^{-10}w \\ -5.4978\,X\,10^{-10}w \end{bmatrix}$$

$$= \begin{bmatrix} 0.0697w \\ 1.2177w \\ -0.10479w \\ -0.06925w \\ -1.21661w \\ -0.20793w \end{bmatrix}$$

It is to be noted that $\{F_m\}$ are the end actions due to joint loads. Hence it must be added to the corresponding end actions in the restrained structure in order to obtain the end actions due to the loads. Therefore, $\{F_m\}_{actual}$ are the true member end actions due to actual loading system can be expressed as

$$\{F_m\}_{actual} = \{F_m\} + \{F_{fm}\}$$

Where, $\{F_{fm}\}$ are the end actions in the restrained structure. Since there is no load acting on member (1), the actual end actions will be:

$$\{F_m\}_{actual} = \begin{bmatrix} 0.0697w \\ 1.2177w \\ -0.10479w \\ -0.06925w \\ -1.21661w \\ -0.20793w \end{bmatrix} + \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} 0.0697w \\ 1.2177w \\ -0.10479w \\ -0.06925w \\ -1.21661w \\ -0.20793w \end{bmatrix}$$

Member (2)

In similar way, the member forces in member (2) will be $\{F_m\}_{(2)} = [K]_{(2)}\{d\}_{(2)}$

$$\begin{bmatrix} F_{x_2} \\ F_{y_2} \\ M_2 \\ F_{x_3} \\ F_{y_3} \\ M_3 \end{bmatrix} = 10^9 \begin{bmatrix} 16 & 0 & 0 & -16 & 0 & 0 \\ 0 & 0.284 & 0.426 & 0 & -0.284 & 0.426 \\ 0 & 0.426 & 0.853 & 0 & -0.426 & 0.426 \\ -16 & 0 & 0 & 16 & 0 & 0 \\ 0 & -0.284 & -0.426 & 0 & 0.284 & -0.426 \\ 0 & 0.426 & 0.426 & 0 & -0.426 & 0.853 \end{bmatrix} \times \begin{bmatrix} 4.327\ X\ 10^{-12}w \\ -1.7127\ X\ 10^{-10}w \\ -5.4978\ X\ 10^{-10}w \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

$$= \begin{bmatrix} 0.069232w \\ -0.28325w \\ -0.54215w \\ -0.06923w \\ 0.283245w \\ -0.3076w \end{bmatrix}$$

The actual member forces in the member (2) will be:

$$\{F_m\}_{actual} = \begin{bmatrix} 0.069232w \\ -0.28325w \\ -0.54215w \\ -0.06923w \\ 0.283245w \\ -0.3076w \end{bmatrix} + \begin{bmatrix} 0 \\ 1.5w \\ 0.75w \\ 0 \\ 1.5w \\ -0.75w \end{bmatrix} = \begin{bmatrix} 0.0692w \\ 1.2167w \\ 0.2078w \\ -0.0692w \\ 1.7832w \\ -1.0576w \end{bmatrix}$$

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

SCHOOL OF MECHANICAL ENGINEERING
DEPARTMENT OF AERONAUTICAL ENGINEERING

UNIT – IV –FINITE ELEMENT METHOD FOR AIRCRAFT STRUCTURES – SAEA1504

UNIT-IV
TWO DIMENSIONAL ELASTICITY AND AXISYMMETRIC ELASTICITY

4.0 CONSTANT STRAIN TRIANGULAR ELEMENT

The triangular elements with different numbers of nodes are used for solving two dimensional solid members. The linear triangular element was the first type of element developed for the finite element analysis of 2D solids. However, it is observed that the linear triangular element is less accurate compared to linear quadrilateral elements. But the triangular element is still a very useful element for its adaptivity to complex geometry. These are used if the geometry of the 2D model is complex in nature. Constant strain triangle (CST) is the simplest element to develop mathematically. In CST, strain inside the element has no variation (Ref. module 3, lecture 2) and hence element size should be small enough to obtain accurate results. As indicated earlier, the displacement is expressed in two orthogonal directions in case of 2D solid elements. Thus the displacement field can be written as

$$\{d\} = \begin{Bmatrix} u \\ v \end{Bmatrix} \tag{4.1.1}$$

Here, *u* and *v* are the displacements parallel to *x* and *y* directions respectively.

4.1.1 Element Stiffness Matrix for CST

A typical triangular element assumed to represent a subdomain of a plane body under plane stress/strain condition is represented in Fig. 4.1.1. The displacement (u, v) of any point P is represented in terms of nodal displacements

$$\begin{aligned} u &= N_1 u_1 + N_2 u_2 + N_3 u_3 \\ v &= N_1 v_1 + N_2 v_2 + N_3 v_3 \end{aligned} \tag{4.1.2}$$

Where, N_1, N_2, N_3 are the shape functions as described in module 3, lecture 2.

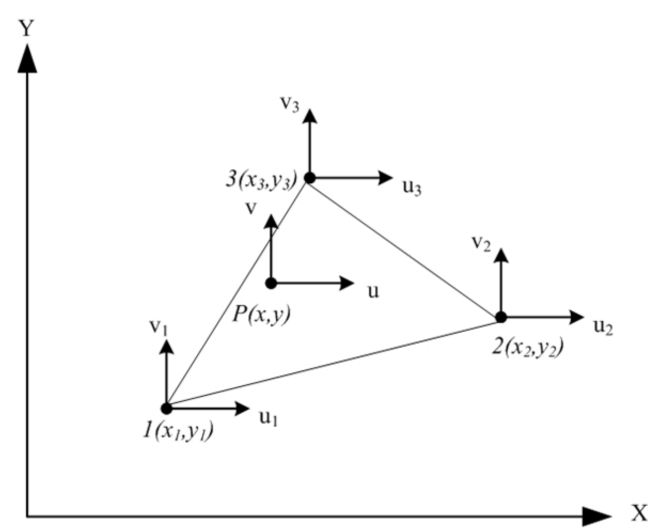


Fig. 4.1.1 Linear triangular element for plane stress/strain

The strain-displacement relationship for two dimensional plane stress/strain problem can be simplified in the following form from three dimensional cases .

$$\begin{aligned}
\varepsilon_x &= \frac{\partial u}{\partial x} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial x}\right)^2\right] \\
\varepsilon_y &= \frac{\partial v}{\partial y} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial y}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2\right] \\
\gamma_{xy} &= \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} + \left[\frac{\partial u}{\partial x}\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\frac{\partial v}{\partial y}\right]
\end{aligned} \tag{4.1.3}$$

In case of small amplitude of displacement, one can ignore the nonlinear term of the above equation and will reach the following expression.

$$\begin{aligned}
\varepsilon_x &= \frac{\partial u}{\partial x} \\
\varepsilon_y &= \frac{\partial v}{\partial y} \\
\gamma_{xy} &= \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}
\end{aligned} \tag{4.1.4}$$

Hence the element strain components can be represented as,

$$\varepsilon = \begin{cases}
\varepsilon_x = \dfrac{\partial u}{\partial x} = \dfrac{\partial N_1}{\partial x}u_1 + \dfrac{\partial N_2}{\partial x}u_2 + \dfrac{\partial N_3}{\partial x}u_3 \\
\varepsilon_y = \dfrac{\partial v}{\partial y} = \dfrac{\partial N_1}{\partial y}v_1 + \dfrac{\partial N_2}{\partial y}v_2 + \dfrac{\partial N_3}{\partial y}v_3 \\
\gamma_{xy} = \dfrac{\partial u}{\partial y} + \dfrac{\partial v}{\partial x} = \dfrac{\partial N_1}{\partial y}u_1 + \dfrac{\partial N_2}{\partial y}u_2 + \dfrac{\partial N_3}{\partial y}u_3 + \dfrac{\partial N_1}{\partial x}v_1 + \dfrac{\partial N_2}{\partial x}v_2 + \dfrac{\partial N_3}{\partial x}v_3
\end{cases}$$

Or,

$$\varepsilon = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix}
\dfrac{\partial N_1}{\partial x} & \dfrac{\partial N_2}{\partial x} & \dfrac{\partial N_3}{\partial x} & 0 & 0 & 0 \\
0 & 0 & 0 & \dfrac{\partial N_1}{\partial y} & \dfrac{\partial N_2}{\partial y} & \dfrac{\partial N_3}{\partial y} \\
\dfrac{\partial N_1}{\partial y} & \dfrac{\partial N_2}{\partial y} & \dfrac{\partial N_3}{\partial y} & \dfrac{\partial N_1}{\partial x} & \dfrac{\partial N_2}{\partial x} & \dfrac{\partial N_3}{\partial x}
\end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ v_1 \\ v_2 \\ v_3 \end{Bmatrix} \tag{4.1.5}$$

Or, $\varepsilon = [B]\{d\}$ (4.1.6)

In the above equation [B] is called as strain displacement relationship matrix. The shape functions for the 3 node triangular element in Cartesian coordinate is represented as,

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

Or,

$$\begin{Bmatrix} N_1 \\ N_2 \\ N_3 \end{Bmatrix} = \begin{Bmatrix} \frac{1}{2A}\left[\alpha_1 + \beta_1 x + \gamma_1 y\right] \\ \frac{1}{2A}\left[\alpha_2 + \beta_2 x + \gamma_2 y\right] \\ \frac{1}{2A}\left[\alpha_3 + \beta_3 x + \gamma_3 y\right] \end{Bmatrix} \tag{4.1.7}$$

Where,

$$\begin{aligned} &\alpha_1 = \left(x_2y_3 - x_3y_2\right), && \alpha_2 = \left(x_3y_1 - x_1y_3\right), && \alpha_3 = \left(x_1y_2 - x_2y_1\right), \\ &\beta_1 = \left(y_2 - y_3\right), && \beta_2 = \left(y_3 - y_1\right), && \beta_3 = \left(y_1 - y_2\right), \\ &\gamma_1 = \left(x_3 - x_2\right), && \gamma_2 = \left(x_2 - x_1\right), && \gamma_3 = \left(x_2 - x_1\right), \end{aligned} \tag{4.1.8}$$

Hence the required partial derivatives of shape functions are,

$$\frac{\partial N_1}{\partial x} = \frac{\beta_1}{2A}, \qquad \frac{\partial N_2}{\partial x} = \frac{\beta_2}{2A}, \qquad \frac{\partial N_3}{\partial x} = \frac{\beta_3}{2A},$$

$$\frac{\partial N_1}{\partial y} = \frac{\gamma_1}{2A}, \qquad \frac{\partial N_2}{\partial x} = \frac{\gamma_2}{2A}, \qquad \frac{\partial N_3}{\partial x} = \frac{\gamma_3}{2A},$$

Hence the value of [B] becomes:

$$[B] = \begin{bmatrix} \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial x} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{\partial N_1}{\partial y} & \frac{\partial N_2}{\partial y} & \frac{\partial N_3}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_2}{\partial y} & \frac{\partial N_3}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial x} \end{bmatrix}$$

Or, $$[B] = \frac{1}{2A}\begin{bmatrix} \beta_1 & \beta_2 & \beta_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & \gamma_1 & \gamma_2 & \gamma_3 \\ \gamma_1 & \gamma_2 & \gamma_3 & \beta_1 & \beta_2 & \beta_3 \end{bmatrix} \tag{4.1.9}$$

According to Variational principle described in module 2, lecture 1, the stiffness matrix is represented as,

$$[k] = \iiint_{\Omega} [B]^T [D][B] \, d\Omega \tag{4.1.10}$$

Since, [B] and [D] are constant matrices; the above expression can be expressed as

$$[k] = [B]^T [D][B] \iiint_{V} dV = [B]^T [D][B] V \tag{4.1.11}$$

For a constant thickness (t), the volume of the element will become $A.t$. Hence the above equation becomes,

$$[k] = [B]^T [D][B] At \tag{4.1.12}$$

For plane stress condition, [D] matrix will become:

$$[D] = \frac{E}{1-\mu^2}\begin{bmatrix} 1 & \mu & 0 \\ \mu & 1 & 0 \\ 0 & 0 & \frac{1-\mu}{2} \end{bmatrix} \tag{4.1.13}$$

Therefore, for a plane stress problem, the element stiffness matrix becomes,

$$[k] = \frac{Et}{4A\left(1-\mu^2\right)}\begin{bmatrix} \beta_1 & 0 & \gamma_1 \\ \beta_2 & 0 & \gamma_2 \\ \beta_3 & 0 & \gamma_3 \\ 0 & \gamma_1 & \beta_1 \\ 0 & \gamma_2 & \beta_2 \\ 0 & \gamma_3 & \beta_3 \end{bmatrix}\begin{bmatrix} 1 & \mu & 0 \\ \mu & 1 & 0 \\ 0 & 0 & \frac{1-\mu}{2} \end{bmatrix}\begin{bmatrix} \beta_1 & \beta_2 & \beta_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & \gamma_1 & \gamma_2 & \gamma_3 \\ \gamma_1 & \gamma_2 & \gamma_3 & \beta_1 & \beta_2 & \beta_3 \end{bmatrix} \tag{4.1.14}$$

Or,

$$[k] = \frac{Et}{4A\left(1-\mu^2\right)}\begin{bmatrix} \beta_1^2 + C\gamma_1^2 & \beta_1\beta_2 + C\gamma_1\gamma_2 & \beta_1\beta_3 + C\gamma_1\gamma_3 & \frac{(1+\mu)}{2}\beta_1\gamma_1 & \mu\beta_1\gamma_2 + C\beta_2\gamma_1 & \mu\beta_1\gamma_3 + C\beta_3\gamma_1 \\ & \beta_2^2 + C\gamma_2^2 & \beta_2\beta_3 + C\gamma_2\gamma_3 & \mu\beta_2\gamma_1 + C\beta_1\gamma_2 & \frac{(1+\mu)}{2}\beta_2\gamma_2 & \mu\beta_2\gamma_3 + C\beta_3\gamma_2 \\ & & \beta_3^2 + C\gamma_3^2 & \mu\beta_3\gamma_1 + C\beta_1\gamma_3 & \mu\beta_3\gamma_2 + C\beta_2\gamma_3 & \frac{(1+\mu)}{2}\beta_3\gamma_3 \\ & & & \gamma_1^2 + C\beta_1^2 & \gamma_1\gamma_2 + C\beta_1\beta_2 & \gamma_1\gamma_3 + C\beta_1\beta_3 \\ & Sym. & & & \gamma_2^2 + C\beta_2^2 & \gamma_2\gamma_3 + C\beta_2\beta_3 \\ & & & & & \gamma_3^2 + C\beta_3^2 \end{bmatrix} \tag{4.1.15}$$

Where, $C = \frac{(1-\mu)}{2}$

Similarly for plane strain condition, [D] matrix is equal to,

$$[D]=\frac{E}{(1+\mu)(1-2\mu)}\begin{bmatrix}(1-\mu) & \mu & 0\\ \mu & (1-\mu) & 0\\ 0 & 0 & \frac{1-2\mu}{2}\end{bmatrix} \tag{4.1.16}$$

Hence the element stiffness matrix will become:

$$[k]=\frac{Et}{2A(1+\mu)}\begin{bmatrix}M\beta_1^2+\gamma_1^2 & M\beta_1\beta_2+\gamma_1\gamma_2 & M\beta_1\beta_3+\gamma_1\gamma_3 & (\mu+1)\beta_1\gamma_1 & \mu\beta_1\gamma_2+\beta_2\gamma_1 & \mu\beta_1\gamma_3+\beta_3\gamma_1\\ & M\beta_2^2+\gamma_2^2 & M\beta_2\beta_3+\gamma_2\gamma_3 & \mu\beta_2\gamma_1+\beta_1\gamma_2 & (\mu+1)\beta_2\gamma_2 & \mu\beta_2\gamma_3+\beta_3\gamma_2\\ & & M\beta_3^2+\gamma_3^2 & \mu\beta_3\gamma_1+\beta_1\gamma_3 & \mu\beta_3\gamma_2+C\beta_2\gamma_3 & (\mu+1)\beta_3\gamma_3\\ & & & M\gamma_1^2+\beta_1^2 & M\gamma_1\gamma_2+\beta_1\beta_2 & M\gamma_1\gamma_3+\beta_1\beta_3\\ & Sym. & & & M\gamma_2^2+\beta_2^2 & M\gamma_2\gamma_3+\beta_2\beta_3\\ & & & & & M\gamma_3^2+\beta_3^2\end{bmatrix} \tag{4.1.17}$$

Where $M=(1-\mu)$

4.1.2 Nodal Load Vector for CST

From the principle of virtual work,

$$\int_\Omega \delta\{\varepsilon\}^T\{\sigma\}d\Omega=\int_\Gamma \delta\{u\}^T\{F_\Gamma\}d\Gamma+\int_\Omega \delta\{u\}^T\{F_\Omega\}d\Omega \tag{4.1.18}$$

Where, F_Γ, and F_Ω are the surface and body forces respectively. Using the relationship between stress-stain and strain displacement, one can derive the following expressions:

$$\{\sigma\}=[D][B]\{d\},\quad \delta\{\varepsilon\}=[B]\delta\{d\}\quad \text{and}\quad \delta\{u\}=[N]\delta\{d\} \tag{4.1.19}$$

Hence eq. (5.1.18) can be rewritten as,

$$\int_\Omega \delta\{d\}^T[B]^T[D][B]\{d\}d\Omega=\int_\Gamma \delta\{d\}^T\left[N^s\right]^T\{F_\Gamma\}d\Gamma+\int_\Omega \delta\{d\}^T[N]^T\{F_\Omega\}d\Omega \tag{4.1.20}$$

Or,
$$\int_\Omega [B]^T[D][B]\{d\}d\Omega=\int_\Gamma \left[N^s\right]^T\{F_\Gamma\}d\Gamma+\int_\Omega [N]^T\{F_\Omega\}d\Omega \tag{4.1.21}$$

Here, [N] is the shape function along the boundary where forces are prescribed. Eq.(4.1.21) is equivalent to $[k]\{d\}=\{F\}$, and thus, the nodal load vector becomes

$$\{F\}=\int_\Gamma \left[N^s\right]^T\{F_\Gamma\}d\Gamma+\int_\Omega [N]^T\{F_\Omega\}d\Omega \tag{4.1.22}$$

For a constant thickness of the triangular element eq.(5.1.22) can be rewritten as

$$\{F\}=t\int_S \left[N^s\right]^T\{F_\Gamma\}ds+t\int_A [N]^T\{F_\Omega\}dA \tag{4.1.23}$$

For the a three node triangular two dimensional element, one can represent F_Ω and F_Γ as,

$$\{F_\Omega\} = \begin{Bmatrix} F_{\Omega x} \\ F_{\Omega y} \end{Bmatrix} \text{ and } \{F_\Gamma\} = \begin{Bmatrix} F_{\Gamma x} \\ F_{\Gamma y} \end{Bmatrix}$$

For example, in case of gravity load on CST element, $\{F_\Omega\} = \begin{Bmatrix} F_{\Omega x} \\ F_{\Omega y} \end{Bmatrix} = \begin{Bmatrix} 0 \\ -\rho g \end{Bmatrix}$

For this case, the shape functions in terms of area coordinates are:

$$[N] = \begin{bmatrix} L_1 & L_2 & L_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & L_1 & L_2 & L_3 \end{bmatrix} \tag{4.1.24}$$

As a result, the force vector on the element considering only gravity load, will become,

$$\{F\} = t\int_A \begin{bmatrix} L_1 & 0 \\ L_2 & 0 \\ L_3 & 0 \\ 0 & L_1 \\ 0 & L_2 \\ 0 & L_3 \end{bmatrix} \begin{Bmatrix} 0 \\ -\rho g \end{Bmatrix} dA = t\int_A \begin{Bmatrix} 0 \\ 0 \\ 0 \\ -L_1\rho g \\ -L_2\rho g \\ -L_3\rho g \end{Bmatrix} dA = -\rho g t \int_A \begin{Bmatrix} 0 \\ 0 \\ 0 \\ L_1 \\ L_2 \\ L_3 \end{Bmatrix} dA \tag{4.1.25}$$

The integration in terms of area coordinate is given by,

$$\int_A L_1^p L_2^q L_3^r dA = \frac{p!\,q!\,r!}{(p+q+r+2)!} 2A \tag{4.1.26}$$

Thus, the nodal load vector will finally become

$$\{F\} = -\rho g t \begin{Bmatrix} 0 \\ 0 \\ 0 \\ \frac{1!0!0!}{(1+0+0+2)!}2A \\ \frac{0!1!0!}{(0+1+0+2)!}2A \\ \frac{0!0!1!}{(0+0+1+2)!}2A \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ -\frac{\rho g A t}{3} \\ -\frac{\rho g A t}{3} \\ -\frac{\rho g A t}{3} \end{Bmatrix} = -\frac{\rho g A t}{3} \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 1 \\ 1 \\ 1 \end{Bmatrix} \tag{4.1.27}$$

FOUR - NODED RECTANGULAR ELEMENT

A typical element is shown in Figure 5.5. It is rectangular and has four nodes, one at each vertex. The nodes have coordinates (X_1, Y_1), (X_2, Y_2), (X_3, Y_3) and (X_4, Y_4) in the global Cartesian coordinate frame (*OXY*) as shown in the figure. For ease of derivation, we define

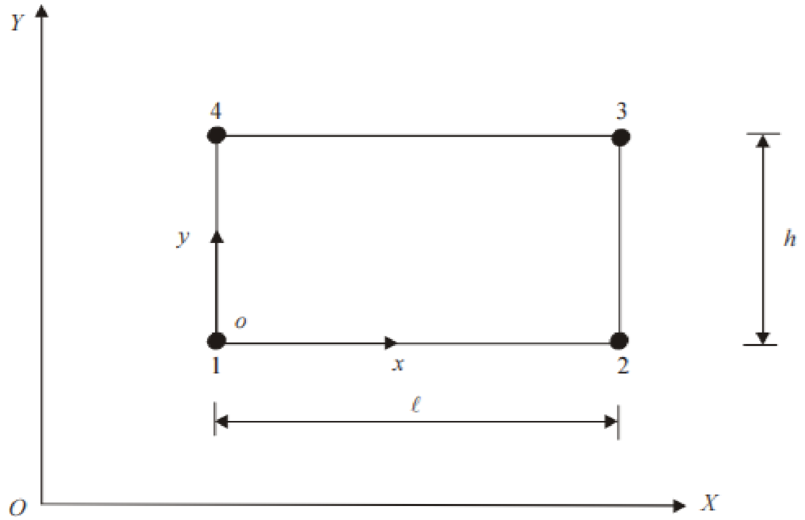


Fig. 5.5 A four-noded rectangular element.

a *local* coordinate frame (*oxy*). Each node has temperature d.o.f. *T* or two d.o.f., viz., *u* and *v*, the translations along the global *X* and *Y* axes, respectively. In what follows, we will derive the shape (interpolation) functions for the temperature field and use the same for the displacement field.

We assume that the temperature field over the element is given by

$$T(x, y) = c_0 + c_1 x + c_2 y + c_3 xy$$

……..4.4.1

We observe that as we have four nodes now, we are able to take a fourth term in our polynomial field. Considering that this expression should reduce to the nodal temperatures at the nodal points we have, for a rectangular element of size ($\ell \times h$) as shown in Figure 5.5,

$$T_1 = c_0, \qquad T_2 = c_0 + c_1\ell,$$

………..4.4.2

$$T_3 = c_0 + c_1\ell + c_2 h + c_3\ell h, \qquad T_4 = c_0 + c_2 h$$

Solving for c_0, c_1, c_2 and c_3, we get

$$c_0 = T_1, \qquad c_1 = \frac{T_2 - T_1}{\ell}, \qquad c_2 = \frac{T_4 - T_1}{h}, \qquad c_3 = \frac{T_3 + T_1 - T_2 - T_4}{\ell h}$$

………4.4.3

$$T(x, y) = \left(1 - \frac{x}{\ell} - \frac{y}{h} + \frac{xy}{\ell h}\right)T_1 + \left(\frac{x}{\ell} - \frac{xy}{\ell h}\right)T_2 + \left(\frac{xy}{\ell h}\right)T_3 + \left(\frac{y}{h} - \frac{xy}{\ell h}\right)T_4$$

........4.4.4

In our standard finite element notation, we write this as

$$T(x,y) = \begin{bmatrix} N_1 & N_2 & N_3 & N_4 \end{bmatrix} \begin{Bmatrix} T_1 \\ T_2 \\ T_3 \\ T_4 \end{Bmatrix}$$

..........4.4.5

Thus we obtain the shape functions N_i:

$$N_1 = 1 - \frac{x}{\ell} - \frac{y}{h} + \frac{xy}{\ell h}, \qquad N_2 = \frac{x}{\ell} - \frac{xy}{\ell h}$$

$$N_3 = \frac{xy}{\ell h}, \qquad N_4 = \frac{y}{h} - \frac{xy}{\ell h}$$

.........4.4.6

If this element is used to model structural mechanics problems, each node will have two d.o.f., viz., u and v, and we can write the displacements at any interior point in terms of the nodal displacements using the same shape functions as follows:

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} = [N]\{\delta\}^e$$

.........4.4.7

4.1.3 Four-node Quadrilateral Element

Consider the parent four-node square element in ξ–η frame as well as the general quadrilateral element in the physical *x-y* space as shown in Figure 5.25. The coordinates of any point *P*(*x*, *y*) are interpolated from nodal coordinates as follows:

$$x = \sum_{i=1}^{4} N_i x_i \qquad 4.1.1$$

$$y = \sum_{i=1}^{4} N_i y_i \qquad 4.1.2$$

where N_i are as given in Eqs.

We observe that this element permits a linear variation of unknown field variable along x = constant or y = constant lines, and thus it is known as the *bilinear element*. Strain and heat flux are not necessarily constant within the element.

We will now discuss a typical higher order element.

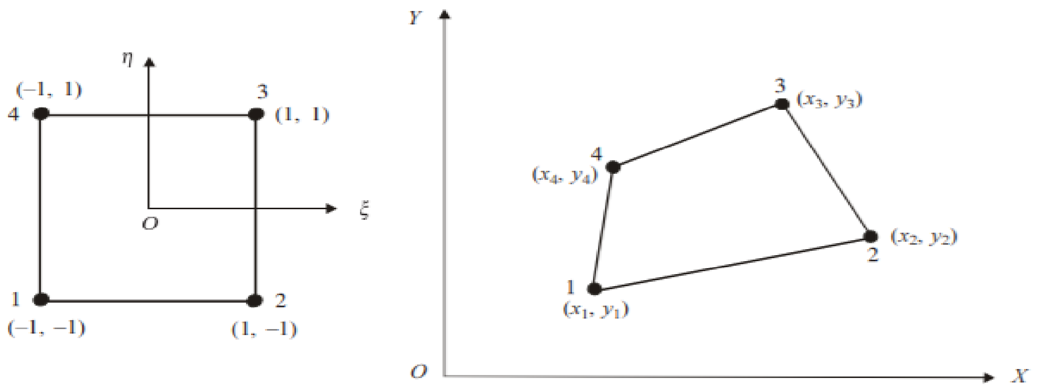


Fig 4.1.3 **A general four-noded quadrilateral element.**

The displacements of the interior point *P* are also interpolated from nodal deflections using the same shape functions for this element, which is therefore called an "isoparametric element". The displacements can be written as

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} \qquad 4.1.3$$

Let us now show that the displacements in this element are compatible across inter-element boundaries. Along edge 2–3 ($\xi = 1$), for example, the shape functions $N_1 = 0 = N_4$. Thus, irrespective of the shape of the element in the physical *x-y* space (i.e. even for a general quadrilateral), the displacements of any point on edge 2–3 are given as

$$u = N_2 u_2 + N_3 u_3$$
$$v = N_2 v_2 + N_3 v_3 \qquad 4.1.4$$

Therefore, we observe that even for a general quadrilateral shape, the isoparametric element guarantees the continuity of displacements across inter-element boundary edges.

Recall our observation that the nodal d.o.f. are in *X-Y* directions and not along ξ–η. Also the ξ-η axes, as transformed into real physical *x–y* space, may not be orthogonal. Thus we write the strain displacement relations in an orthogonal, Cartesian frame of reference, as

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \dfrac{\partial u}{\partial x} \\ \dfrac{\partial v}{\partial y} \\ \dfrac{\partial u}{\partial y} + \dfrac{\partial v}{\partial x} \end{Bmatrix} \qquad 4.1.5$$

We observe that u and v are given in terms of shape functions N_i which are expressed in the ξ–η coordinates rather than *x–y*. For a general function *f*, using the chain rule of differentiation, we obtain the equations

$$\frac{\partial f}{\partial x} = \frac{\partial f}{\partial \xi}\frac{\partial \xi}{\partial x} + \frac{\partial f}{\partial \eta}\frac{\partial \eta}{\partial x}$$
$$\frac{\partial f}{\partial y} = \frac{\partial f}{\partial \xi}\frac{\partial \xi}{\partial y} + \frac{\partial f}{\partial \eta}\frac{\partial \eta}{\partial y} \qquad 4.1.6$$

it is easier to write down derivatives of (x–y) in terms of (ξ–η) rather than the other way round. Thus we write

$$\frac{\partial f}{\partial \xi} = \frac{\partial f}{\partial x}\frac{\partial x}{\partial \xi} + \frac{\partial f}{\partial y}\frac{\partial y}{\partial \xi}$$
$$\frac{\partial f}{\partial \eta} = \frac{\partial f}{\partial x}\frac{\partial x}{\partial \eta} + \frac{\partial f}{\partial y}\frac{\partial y}{\partial \eta} \qquad 4.1.7$$

Thus, in general, we can write

$$\begin{Bmatrix} \dfrac{\partial}{\partial \xi} \\ \dfrac{\partial}{\partial \eta} \end{Bmatrix} = \begin{bmatrix} \dfrac{\partial x}{\partial \xi} & \dfrac{\partial y}{\partial \xi} \\ \dfrac{\partial x}{\partial \eta} & \dfrac{\partial y}{\partial \eta} \end{bmatrix} \begin{Bmatrix} \dfrac{\partial}{\partial x} \\ \dfrac{\partial}{\partial y} \end{Bmatrix} \qquad 4.1.8$$

i.e.

$$\begin{Bmatrix} \dfrac{\partial}{\partial \xi} \\ \dfrac{\partial}{\partial \eta} \end{Bmatrix} = [J] \begin{Bmatrix} \dfrac{\partial}{\partial x} \\ \dfrac{\partial}{\partial y} \end{Bmatrix} \qquad 4.1.9$$

where

$$[J] = \begin{bmatrix} \dfrac{\partial x}{\partial \xi} & \dfrac{\partial y}{\partial \xi} \\ \dfrac{\partial x}{\partial \eta} & \dfrac{\partial y}{\partial \eta} \end{bmatrix} \qquad 4.1.10$$

$$[J] = \begin{bmatrix} \sum \dfrac{\partial N_i}{\partial \xi} x_i & \sum \dfrac{\partial N_i}{\partial \xi} y_i \\ \sum \dfrac{\partial N_i}{\partial \eta} x_i & \sum \dfrac{\partial N_i}{\partial \eta} y_i \end{bmatrix} \qquad 4.1.11$$

Here, [*J*] is known as the Jacobian matrix relating the derivatives in two coordinate frames. This generic expression for [*J*] is true for all 2-d elements, but the actual coefficients in the matrix will depend on the shape functions being employed in a given element and the nodal coordinates.

For this particular element, for example, the Jacobian is

$$[J] = \begin{bmatrix} \left(\dfrac{1-\eta}{4}\right)(x_2 - x_1) + \left(\dfrac{1+\eta}{4}\right)(x_3 - x_4) & \left(\dfrac{1-\eta}{4}\right)(y_2 - y_1) + \left(\dfrac{1+\eta}{4}\right)(y_3 - y_4) \\ \left(\dfrac{1-\xi}{4}\right)(x_4 - x_1) + \left(\dfrac{1+\xi}{4}\right)(x_3 - x_2) & \left(\dfrac{1-\xi}{4}\right)(y_4 - y_1) + \left(\dfrac{1+\xi}{4}\right)(y_3 - y_2) \end{bmatrix} \qquad 4.1.12$$

From Eq. (5.155),

$$\begin{Bmatrix} \dfrac{\partial}{\partial x} \\ \dfrac{\partial}{\partial y} \end{Bmatrix} = [J]^{-1} \begin{Bmatrix} \dfrac{\partial}{\partial \xi} \\ \dfrac{\partial}{\partial \eta} \end{Bmatrix} = \frac{1}{|J|} \begin{bmatrix} J_{22} & -J_{12} \\ -J_{21} & J_{11} \end{bmatrix} \begin{Bmatrix} \dfrac{\partial}{\partial \xi} \\ \dfrac{\partial}{\partial \eta} \end{Bmatrix} \qquad 4.1.13$$

We can now obtain the required strain displacement relations.

$$\{\varepsilon\} = \begin{Bmatrix} \dfrac{\partial u}{\partial x} \\ \dfrac{\partial v}{\partial y} \\ \dfrac{\partial u}{\partial y} + \dfrac{\partial v}{\partial x} \end{Bmatrix} = \underbrace{\begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 \\ 0 & 1 & 1 & 0 \end{bmatrix}}_{[B_1]} \begin{Bmatrix} \dfrac{\partial u}{\partial x} \\ \dfrac{\partial u}{\partial y} \\ \dfrac{\partial v}{\partial x} \\ \dfrac{\partial v}{\partial y} \end{Bmatrix} \qquad 4.1.14$$

$$\begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial u}{\partial y} \\ \frac{\partial v}{\partial x} \\ \frac{\partial v}{\partial y} \end{Bmatrix} = \underbrace{\begin{bmatrix} \frac{J_{22}}{|J|} & -\frac{J_{12}}{|J|} & 0 & 0 \\ -\frac{J_{21}}{|J|} & \frac{J_{11}}{|J|} & 0 & 0 \\ 0 & 0 & \frac{J_{22}}{|J|} & -\frac{J_{12}}{|J|} \\ 0 & 0 & \frac{-J_{21}}{|J|} & \frac{J_{11}}{|J|} \end{bmatrix}}_{[B_2]} \begin{Bmatrix} \frac{\partial u}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \\ \frac{\partial v}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \end{Bmatrix} \qquad 4.1.15$$

From Eq. (5.149),

$$\begin{Bmatrix} \frac{\partial u}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \\ \frac{\partial v}{\partial \xi} \\ \frac{\partial v}{\partial \eta} \end{Bmatrix} = \begin{bmatrix} \frac{\partial N_1}{\partial \xi} & 0 & \frac{\partial N_2}{\partial \xi} & 0 & \frac{\partial N_3}{\partial \xi} & 0 & \frac{\partial N_4}{\partial \xi} & 0 \\ \frac{\partial N_1}{\partial \eta} & 0 & \frac{\partial N_2}{\partial \eta} & 0 & \frac{\partial N_3}{\partial \eta} & 0 & \frac{\partial N_4}{\partial \eta} & 0 \\ 0 & \frac{\partial N_1}{\partial \xi} & 0 & \frac{\partial N_2}{\partial \xi} & 0 & \frac{\partial N_3}{\partial \xi} & 0 & \frac{\partial N_4}{\partial \xi} \\ 0 & \frac{\partial N_1}{\partial \eta} & 0 & \frac{\partial N_2}{\partial \eta} & 0 & \frac{\partial N_3}{\partial \eta} & 0 & \frac{\partial N_4}{\partial \eta} \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} \qquad 4.1.16$$

$$= [B_3]\ \{\delta\}^e$$

For this element, for example, the shape function derivatives in the ξ-η frame are as follows:

$$\begin{aligned} \frac{\partial N_1}{\partial \xi} &= \frac{-(1-\eta)}{4}, & \frac{\partial N_1}{\partial \xi} &= \frac{-(1-\eta)}{4} \\ \frac{\partial N_2}{\partial \xi} &= \frac{1-\eta}{4}, & \frac{\partial N_2}{\partial \eta} &= \frac{-(1+\xi)}{4} \\ \frac{\partial N_3}{\partial \xi} &= \frac{1+\eta}{4}, & \frac{\partial N_3}{\partial \eta} &= \frac{(1+\xi)}{4} \\ \frac{\partial N_4}{\partial \xi} &= \frac{-(1+\eta)}{4}, & \frac{\partial N_4}{\partial \eta} &= \frac{(1-\xi)}{4} \end{aligned} \qquad 4.1.17$$

Thus, finally, combining Eqs. 4.1.16 & 4.1.17, the strain-displacement relations can be written as

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_{xy} \end{Bmatrix} = [B]\{\delta\}^e \qquad 4.1.18$$

where $[B] = [B_1][B_2][B_3]$.

The stress-strain relation matrix $[D]$ remains as given earlier we have the element stiffness matrix

$$[k]^e_{8\times 8} = \int_v [B]^T [D][B]\, dv = \iint [B]^T [D][B]\, t\, dx\, dy$$
$$= \int_{-1}^{1} \int_{-1}^{1} [B]^T [D][B] t |J|\, d\xi\, d\eta \qquad 4.1.19$$

where the coordinate transformation has been taken into account and the integrals need now be evaluated over the parent square element in natural coordinates. Since $[B]$ in general varies from point to point within the element, we look for ways of evaluating these integrals numerically within a computer program rather than attempting to derive them explicitly. We will discuss numerical integration schemes in Section

4.1.4 Eight-node Quadrilateral Element

Consider the parent eight-node rectangular element in ξ-η frame as well as the general quadrilateral element in the physical x-y space as shown in Figure 5.26. The coordinates of any point $P(x, y)$ are interpolated from nodal coordinates as follows:

$$x = \sum N_i x_i, \qquad y = \sum N_i y_i \qquad 4.1.20$$

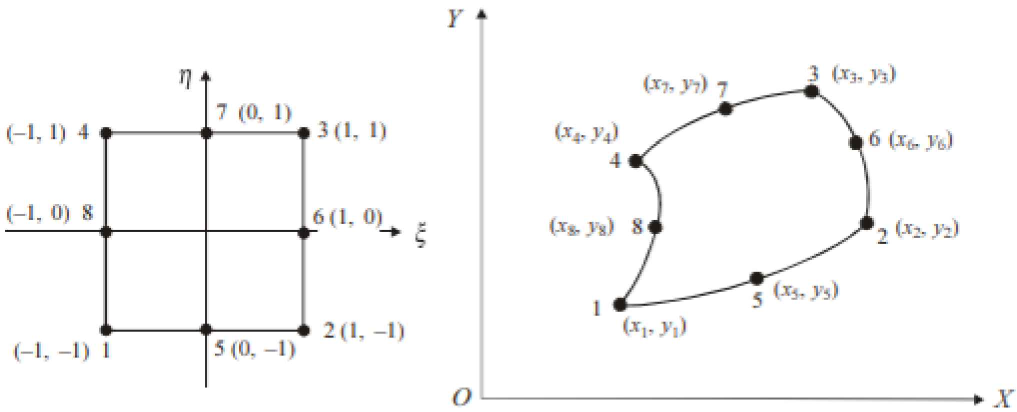


Fig 4.1.4 **A general eight-noded quadrilateral element.**

where we may take coordinates of just the four vertices and follow a simple linear interpolation for coordinate transformation (or) use the full eight nodes to permit curved edges. The displacements of the interior point P are also interpolated from nodal deflections using the same shape functions. Three possibilities arise as shown below:

Coordinate Interpolation	**Displacement Interpolation**
Linear (i.e. four vertex nodes)	Quadratic (i.e. all eight nodes)
Quadratic	Quadratic
Quadratic	Linear

The first element belongs to the category of sub-parametric elements and is useful when the structural geometry is simple polygonal, but unknown field variation may involve sharp variations. The second element is an isoparametric element and permits curved edge modelling as well as quadratic variations in displacements. If the geometry has certain curved features but is in a low stress region, we can use the third element belonging to the superparametric category. The displacements can, in general, be written as

$$\begin{Bmatrix} u \\ v \end{Bmatrix}_{2\times1} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & \cdots \\ 0 & N_1 & 0 & N_2 & \cdots \end{bmatrix}_{2\times16} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ \vdots \end{Bmatrix}_{16\times1} \qquad 4.1.21$$

where the shape functions are appropriately taken from Eqs. 4.1.21 and sizes of the matrices shown are valid for the sub- and isoparametric elements.

The corresponding strain-displacement relation, following Eq. 4.1.21, is written as

$$\{\varepsilon\}_{3\times1} = \underbrace{[B_1]_{3\times4}[B_2]_{4\times4}[B_3]_{4\times16}}\ \{\delta\}^e_{16\times1}$$

$$= [B]_{3\times16}\{\delta\}^e_{16\times1} \qquad 4.1.22$$

For the eight-noded isoparametric element, for example, the Jacobian can be obtained as

$$[J] = \begin{bmatrix} \sum_{i=1}^{8} \frac{\partial N_i}{\partial \xi} x_i & \sum_{i=1}^{8} \frac{\partial N_i}{\partial \xi} y_i \\ \sum_{i=1}^{8} \frac{\partial N_i}{\partial \eta} x_i & \sum_{i=1}^{8} \frac{\partial N_i}{\partial \eta} y_i \end{bmatrix} \qquad 4.1.23$$

The necessary shape function derivatives in the ξ-η frame are summarised now from Eqs. 4.1.22 & 4.1.23.

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

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

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

$$\frac{\partial N_4}{\partial \xi} = \frac{(1+\eta)(2\xi-\eta)}{4}, \qquad \frac{\partial N_4}{\partial \eta} = \frac{(1-\xi)(2\eta-\xi)}{4} \qquad 4.1.24$$

$$\frac{\partial N_5}{\partial \xi} = -\xi(1-\eta), \qquad \frac{\partial N_5}{\partial \eta} = -\frac{(1-\xi^2)}{2}$$

$$\frac{\partial N_6}{\partial \xi} = \frac{1-\eta^2}{2}, \qquad \frac{\partial N_6}{\partial \eta} = -\eta(1+\xi)$$

$$\frac{\partial N_7}{\partial \xi} = -\xi(1+\eta), \qquad \frac{\partial N_7}{\partial \eta} = \frac{(1-\xi^2)}{2}$$

$$\frac{\partial N_8}{\partial \xi} = \frac{-(1-\eta^2)}{2}, \qquad \frac{\partial N_8}{\partial \eta} = -\eta(1-\xi)$$

We can write the expression for the element stiffness matrix as

$$\begin{aligned}[k]^e &= \int_v [B]^T[D][B]\,dv \\ &= \iint [B]^T[D][B]\,t\,dx\,dy \qquad 4.1.25 \\ &= \int_{-1}^{1}\int_{-1}^{1} [B]^T[D][B]\,t\,|J|\,d\xi\,d\eta\end{aligned}$$

which can be evaluated by using suitable numerical integration schemes.

4.2.1 Axisymmetric Element

Many three-dimensional problems show symmetry about an axis of rotation. If the problem geometry is symmetric about an axis and the loading and boundary conditions are symmetric about the same axis, the problem is said to be axisymmetric. Such three-dimensional problems can be solved using two-dimensional finite elements. The axisymmetric problem are most conveniently defined by polar coordinate system with coordinates (r, θ, z) as shown in Fig. 4.2.1. Thus, for axisymmetric analysis, following conditions are to be satisfied.

1. The domain should have an axis of symmetry and is considered as *z* axis.
2. The loadings on the domain has to be symmetric about the axis of revolution, thus they are independent of circumferential coordinate θ.
3. The boundary condition and material properties are symmetric about the same axis and will be independent of circumferential coordinate.

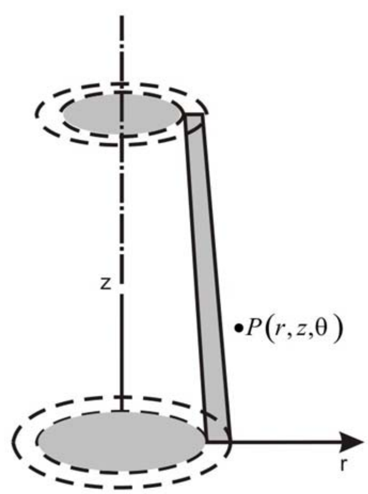


Fig. 4.2.1 Cylindrical coordinates

Axisymmetric solids are of total symmetry about the axis of revolution (i.e., *z*-axis), the field variables, such as the stress and deformation is independent of rotational angle θ. Therefore, the field variables can be defined as a function of (*r*,*z*) and hence the problem becomes a two dimensional problem similar to those of plane stress/strain problems. Axisymmetric problems includes, circular cylinder loaded with uniform external or internal pressure, circular water tank, pressure vessels, chimney, boiler, circular footing resting on soil mass, etc.

4.2.2 Relation between Strain and Displacement

An axisymmetric problem is readily described in cylindrical polar coordinate system: *r*, *z* and θ. Here, θ measures the angle between the plane containing the point and the axis of the coordinate system. At θ = 0, the radial and axial coordinates coincide with the global Cartesian X and Y coordinates. Fig. 4.2.2 shows a cylindrical coordinate system and the definition of the position vectors. Let $\hat{r}, \hat{z}$ and $\hat{\theta}$ be unit vectors in the radial, axial, and circumferential directions at a point in the cylindrical coordinate system.

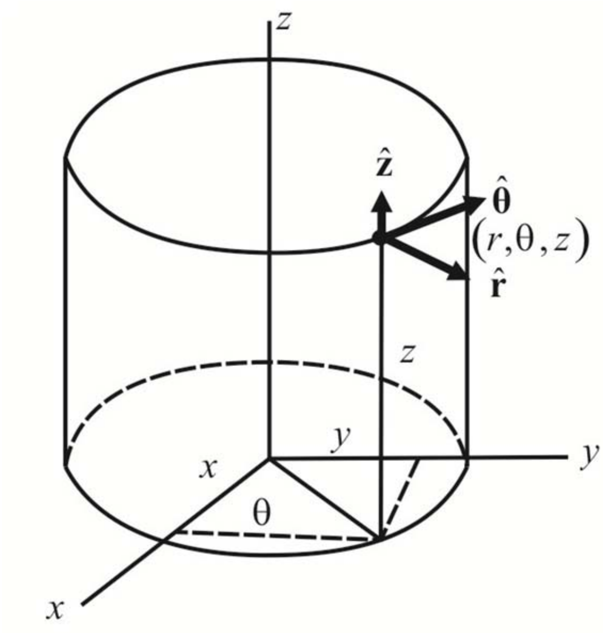
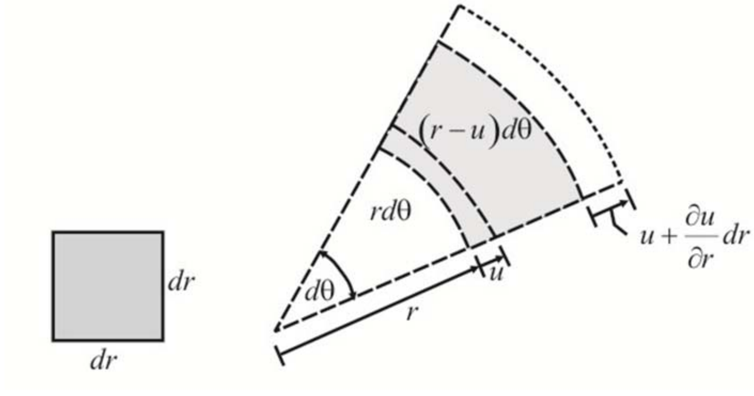


Fig. 4.2.2 Cylindrical Coordinate System

If the loading consists of radial and axial components that are independent of θ and the material is either isotropic or orthotropic and the material properties are independent of θ, the displacement at any point will only have radial (u_r) and axial (u_z) components. The only stress components that will be nonzero are $\sigma_{rr},\ \sigma_{zz},\ \sigma_{\theta\theta}\ and\ \tau_{rz}$.



(a) Element in r-z plane (b) Element in r-θ plane

Fig. 4.2.3 Deformation of the axisymmetric element

A differential element of the body in the r-z plane is shown in Fig. 4.2.3(a). The element undergoes deformation in the radial direction. Therefore, it initiates increase in circumference and associated circumferential strain. Let denote the radial displacement as *u*, the circumferential displacement as *v*, and the axial displacement as *w*. Dashed line represents the deformed positions of the body in Fig. 4.2.3(b). The radial strain can be calculated from the above diagram as

$$\varepsilon_r = \frac{1}{dr}\left(u + \frac{\partial u}{\partial r} \times dr - u\right) = \frac{\partial u}{\partial r} \qquad (4.2.1)$$

Since the rz plane is effectively the same as a rectangular coordinate system, the axial strain will become

$$\varepsilon_z = \frac{1}{dz}\left(w + \frac{\partial w}{\partial z} \times dz - w\right) = \frac{\partial w}{\partial z} \qquad (4.2.2)$$

Considering the original arc length versus the deformed arc length, the differential element undergoes an expansion in the circumferential direction. Before deformation, let the arc length is assumed as d*s* = *r*dθ. After deformation, the arc length will become d*s* = (*r*+*u*) dθ. Thus, the tangential strain will be

$$\varepsilon_\theta = \frac{(r+u)d\theta - rd\theta}{rd\theta} = \frac{u}{r} \qquad (4.2.3)$$

Similarly, the shear strain will be

$$\gamma_{rz} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial r} \qquad (4.2.4)$$

$$\gamma_{r\theta} = 0 \text{ and } \gamma_{z\theta} = 0$$

Thus, there are four strain components present in this case and is given by

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_r \\ \varepsilon_z \\ \varepsilon_\theta \\ \gamma_{rz} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial r} \\ \frac{\partial w}{\partial z} \\ \frac{u}{r} \\ \frac{\partial u}{\partial z} + \frac{\partial w}{\partial r} \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial r} & 0 \\ 0 & \frac{\partial}{\partial z} \\ \frac{1}{r} & 0 \\ \frac{\partial}{\partial z} & \frac{\partial}{\partial r} \end{bmatrix} \begin{Bmatrix} u \\ w \end{Bmatrix} \qquad (4.2.5)$$

4.2.3 Relation between Stress and Strain

The stress strain relation for axisymmetric case can be derived from the three dimensional constitutive relations. We know the stress-strain relation for a three-dimensional solid is

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \frac{E}{(1+\mu)(1-2\mu)} \begin{bmatrix} 1-\mu & \mu & \mu & 0 & 0 & 0 \\ \mu & 1-\mu & \mu & 0 & 0 & 0 \\ \mu & \mu & 1-\mu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\mu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\mu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\mu}{2} \end{bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \nu_{xy} \\ \nu_{yz} \\ \nu_{zx} \end{Bmatrix} \tag{4.2.6}$$

The stresses acting on a differential volume of an axisymmetric solid under axisymmetric loading is shown in Fig. 4.2.4.

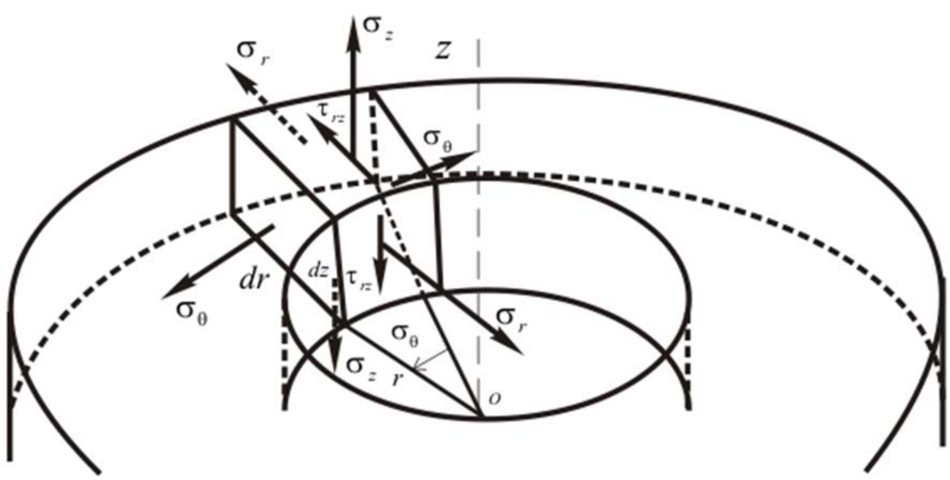


Fig. 4.2.4 Stresses acting on a differential volume

Now, comparing the stress-strain components present in the axisymmetric case, the stress-strain relation can be expressed from the above expression as follows

$$\begin{Bmatrix} \sigma_r \\ \sigma_z \\ \sigma_\theta \\ \tau_{rz} \end{Bmatrix} = \frac{E}{(1+\mu)(1-2\mu)} \begin{bmatrix} 1-\mu & \mu & \mu & 0 \\ \mu & 1-\mu & \mu & 0 \\ \mu & \mu & 1-\mu & 0 \\ 0 & 0 & 0 & \frac{1-2\mu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_r \\ \varepsilon_z \\ \varepsilon_\theta \\ \nu_{rz} \end{Bmatrix} \tag{4.2.7}$$

Thus, the constitutive matrix [D] for the axisymmetric elastic solid will be

$$[D] = \frac{E}{(1+\mu)(1-2\mu)}\begin{bmatrix} 1-\mu & \mu & \mu & 0 \\ \mu & 1-\mu & \mu & 0 \\ \mu & \mu & 1-\mu & 0 \\ 0 & 0 & 0 & \frac{1-2\mu}{2} \end{bmatrix} \tag{4.2.8}$$

4.2.4 Axisymmetric Shell Element

A cylindrical liquid storage container like structures (Fig. 4.2.5) may be idealized using axisymmetric shell element for the finite element analysis. It may be noted that the liquid in the container may be idealized with two dimensional axisymmetric elements. Let us consider the radius, height and, thickness of the circular tank are *R*, *H* and *h* respectively.

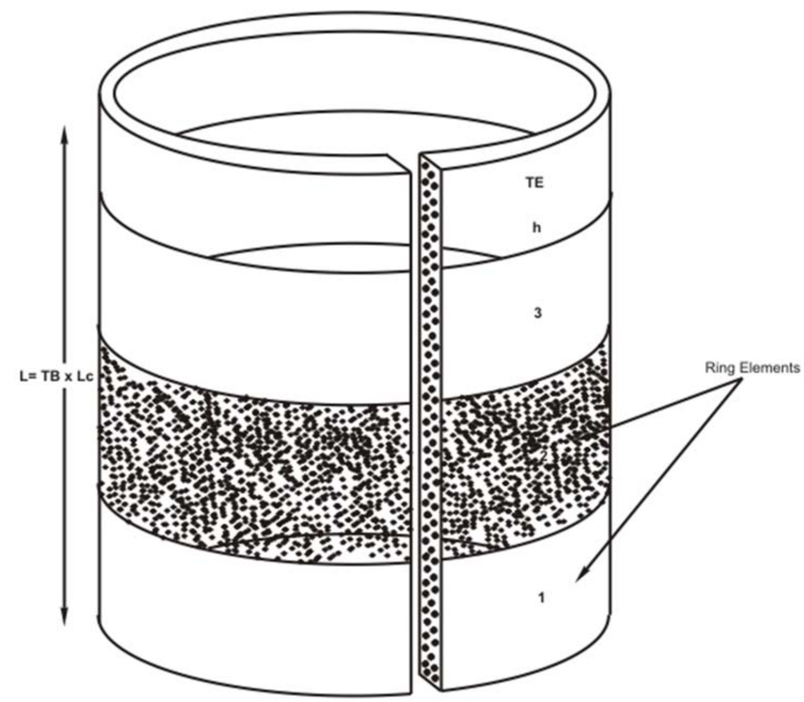


Fig. 4.2.5 Thin wall cylindrical container

The strain energy of the axisymmetric shell element (Fig. 4.2.6) including the effect of both stretching and bending are expressed as

$$U = \frac{1}{2}\int_0^H \left(N_y \varepsilon_y + N_\theta \varepsilon_\theta + M_y \chi_y\right) 2\pi R dy \tag{4.2.9}$$

Here, N_y and N_θ are the membrane force resultants and M_y is the bending moment resultant. The shell is assumed to be linearly elastic, homogeneous and isotropic. Thus the force and moment resultants can be expressed in terms of the mid-surface change in curvature χ_y as follows.

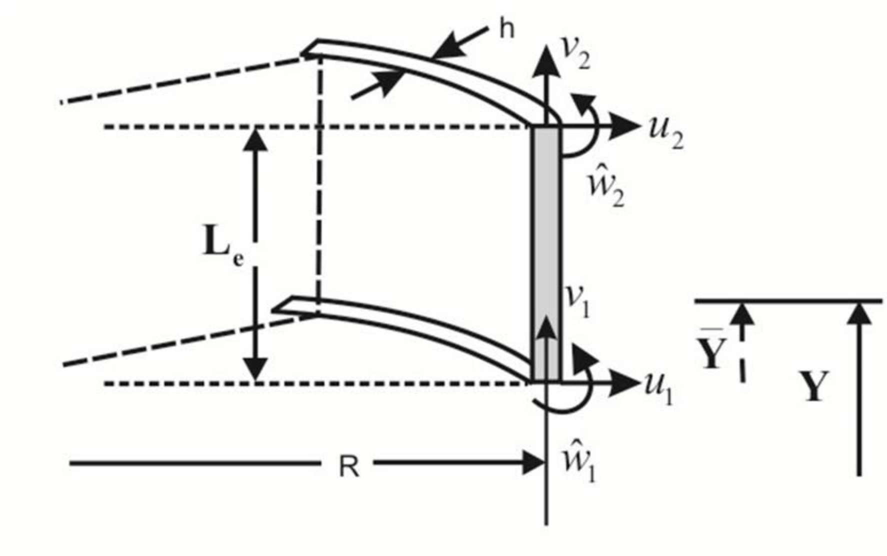


Fig 4.2.6 Axisymmetric plate element

Here, the strain-displacement relation is given by

$$\{\sigma\} = [D]\{\varepsilon\} \tag{4.2.10}$$

In which,

$$\{\sigma\} = \begin{Bmatrix} N_y \\ N_\theta \\ M_y \end{Bmatrix}, \ \{\varepsilon\} = \begin{Bmatrix} \varepsilon_y \\ \varepsilon_\theta \\ \chi_y \end{Bmatrix} \text{ and } [D] = \frac{Eh}{1-\mu^2}\begin{bmatrix} 1 & \mu & 0 \\ \mu & 1 & 0 \\ 0 & 0 & \dfrac{h^2}{12} \end{bmatrix} \tag{4.2.11}$$

The generalized strain vector can be expressed in terms of the displacement vectors as follows.

$$\{\varepsilon\} = [B]\{d\} \tag{4.2.12}$$

Where,

$$\{d\} = \begin{Bmatrix} u \\ v \end{Bmatrix} \text{ and } [B] = \begin{bmatrix} 0 & \dfrac{\partial}{\partial y} \\ \dfrac{1}{R} & 0 \\ -\dfrac{\partial^2}{\partial y^2} & 0 \end{bmatrix} \qquad (4.2.13)$$

Here, u and v are the displacement components in two perpendicular directions. With the use of stress and strain vectors, the potential energy expression are written in terms of displacement vectors as

$$U = \frac{1}{2} \times 2\pi R \int_0^H \left(\{d\}^T [B]^T [D][B]\{d\} \right) dy \qquad (4.2.14)$$

Thus, the element stiffness are derived as

$$[k] = 2\pi R \int_0^H [B]^T [D][B] dy \qquad (4.2.15)$$

Similarly, neglecting the rotary inertia, the kinetic energy can be expressed as

$$T = \frac{1}{2} \times 2\pi R \int_0^H \left(\{\dot{d}\}^T [N]^T m [N] \{\dot{d}\} \right) dy \qquad (4.2.16)$$

Where, m denotes the mass of the shell element per unit area and $\{\dot{d}\}$ represents the velocity vector. Thus, the element mass matrix is given by

$$[M] = 2\pi Rm \int_0^{L_e} [N]^T [N] dy \qquad (4.2.17)$$

Finite Element Formulation of Axisymmetric Element

Finite element formulation for the axisymmetric problem will be similar to that of the two dimensional solid elements. As the field variables, such as the stress and strain is independent of rotational angle θ, circumferential displacement will not appear. Thus, the displacement field variables are expressed as

$$u(r,z) = \sum_{i=1}^{n} N_i(r,z) u_i$$
$$w(r,z) = \sum_{i=1}^{n} N_i(r,z) w_i \qquad (4.3.1)$$

Here, u_i and w_i represent radial and axial displacements respectively at nodes. $N_i(r, z)$ are the shape functions. As the geometry and field variables are independent of rotational angle θ, the interpolation function $N_i(r, z)$ can be expressed similar to 2-dimensional problems by replacing the x and y terms with r and z terms respectively.

4.3.1 Stiffness Matrix of a Triangular Element

Fig. 4.3.1 shows the cylindrical coordinates of a three node triangular element. Hence the analysis of the axisymmetric element can be approached in a similar way as the CST element. Thus the field variables of such an element can be expressed as

$$\begin{aligned} u &= \alpha_0 + \alpha_1 r + \alpha_2 z \\ w &= \alpha_3 + \alpha_4 r + \alpha_5 z \end{aligned} \tag{4.3.2}$$

Or,

$$\{d\} = [\phi]\{\alpha\} \tag{4.3.3}$$

Where,

$$\{d\} = \begin{Bmatrix} u \\ w \end{Bmatrix},\ [\phi] = \begin{bmatrix} 1 & r & z & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & r & z \end{bmatrix} \text{ and } \{\alpha\}^T = \{\alpha_0 \quad \alpha_1 \quad \alpha_2 \quad \alpha_3 \quad \alpha_4 \quad \alpha_5\}$$

Using end conditions,

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ w_1 \\ w_2 \\ w_3 \end{Bmatrix} = \begin{bmatrix} 1 & r_i & z_i & 0 & 0 & 0 \\ 1 & r_j & z_j & 0 & 0 & 0 \\ 1 & r_k & z_j & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & r_i & z_i \\ 0 & 0 & 0 & 1 & r_j & z_j \\ 0 & 0 & 0 & 1 & r_k & z_k \end{bmatrix} \begin{Bmatrix} \alpha_0 \\ \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \\ \alpha_5 \end{Bmatrix} \tag{4.3.4}$$

Or,

$$\begin{aligned} &\{\bar{d}\} = [A]\{\alpha\} \\ &\Rightarrow \{\alpha\} = [A]^{-1}\{\bar{d}\} \end{aligned} \tag{4.3.5}$$

Here $\{\bar{d}\}$ are the nodal displacement vectors.

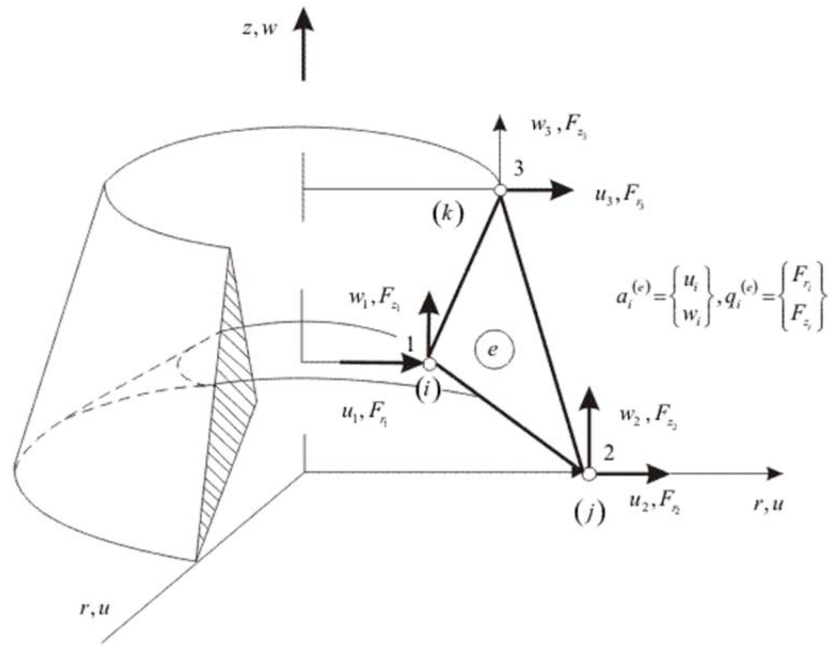


Fig. 4.3.1 Axisymmetric three node triangle in cylindrical coordinates

Putting above values in eq.(4.3.3), the following relations will be obtained.

$$\{d\} = [\phi][A]^{-1}\{\overline{d}\} = [N]\{\overline{d}\} \tag{4.3.6}$$

Or,

$$\{d\} = \begin{Bmatrix} u \\ w \end{Bmatrix} = \begin{bmatrix} N_i & N_j & N_k & 0 & 0 & 0 \\ 0 & 0 & 0 & N_i & N_j & N_k \end{bmatrix} \begin{Bmatrix} r_1 \\ r_2 \\ r_3 \\ z_1 \\ z_2 \\ z_3 \end{Bmatrix} \tag{4.3.7}$$

Using a similar approach as in case of CST elements, the three shape functions $[N_1, N_2, N_3]$ can be assumed as,

$$N_1(r,z) = \frac{1}{2A}\left[(r_2 z_3 - r_3 z_2) + (z_2 - z_3) r + (r_3 - r_2) z\right]$$
$$N_2(r,z) = \frac{1}{2A}\left[(r_3 z_1 - r_1 z_3) + (z_3 - z_1) r + (r_1 - r_3) z\right]$$
$$N_3(r,z) = \frac{1}{2A}\left[(r_1 z_2 - r_2 z_1) + (z_1 - z_2) r + (r_2 - r_1) z\right]$$

Or,

$$N_i(r,z)=\frac{1}{2A}\left(\alpha_i+r\beta_i+z\gamma_i\right)$$
$$N_j(r,z)=\frac{1}{2A}\left(\alpha_j+r\beta_j+z\gamma_j\right) \qquad (4.3.8)$$
$$N_k(r,z)=\frac{1}{2A}\left(\alpha_k+r\beta_k+z\gamma_k\right)$$

Where,

$$\alpha_i=r_jz_k-r_kz_j \qquad \alpha_j=r_kz_i-r_iz_k \qquad \alpha_k=r_iz_j-r_jz_i$$
$$\beta_i=z_j-z_k \qquad \beta_j=z_k-z_i \qquad \beta_k=z_i-z_j$$
$$\gamma_i=r_k-r_j \qquad \gamma_i=r_i-r_k \qquad \gamma_i=r_j-r_i \qquad (4.3.9)$$
$$2A=\frac{1}{2}\left(r_iz_j+r_jz_k+r_kz_i-r_iz_k-r_jz_i-r_kz_j\right)$$

Putting the value of {u,w} in eq. (4.3.7) from eq. (4.3.5),

$$\{\varepsilon\}=\begin{bmatrix} \frac{\partial N_i}{\partial r} & \frac{\partial N_j}{\partial r} & \frac{\partial N_k}{\partial r} & 0 & 0 & 0 \\ \frac{N_i}{r} & \frac{N_j}{r} & \frac{N_k}{r} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{\partial N_i}{\partial z} & \frac{\partial N_j}{\partial z} & \frac{\partial N_k}{\partial z} \\ \frac{\partial N_i}{\partial z} & \frac{\partial N_j}{\partial z} & \frac{\partial N_k}{\partial z} & \frac{\partial N_i}{\partial r} & \frac{\partial N_j}{\partial r} & \frac{\partial N_k}{\partial r} \end{bmatrix}\begin{Bmatrix} r_1 \\ r_2 \\ r_3 \\ z_1 \\ z_2 \\ z_3 \end{Bmatrix}=[B]\{\overline{d}\} \qquad (4.3.10)$$

Thus, the strain displacement matrix can be expressed as,

$$[B]=\frac{1}{2A}\begin{bmatrix} \beta_i & \beta_j & \beta_k & 0 & 0 & 0 \\ \frac{N_i}{r} & \frac{N_j}{r} & \frac{N_k}{r} & 0 & 0 & 0 \\ 0 & 0 & 0 & \gamma_i & \gamma_j & \gamma_k \\ \gamma_i & \gamma_j & \gamma_k & \beta_i & \beta_j & \beta_k \end{bmatrix} \qquad (4.3.11)$$

Where, $r=\frac{r_i+r_j+r_k}{3}$. Thus the stiffness matrix will become

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

Or, $$[k]=\iint\int_0^{2\pi}[B]^T[D][B]\,r\,d\theta\,dA=2\pi\iint[B]^T[D][B]\,r\,dr\,dz \qquad (4.3.12)$$

Since, the term [B] is dependent of 'r' terms; the term $[B]^T[D][B]$ cannot be taken out of integration. Yet, a reasonably accurate solution can be obtained by evaluating the [B] (denoted as [$\underline{B}$]) matrix at the centroid.

Hence, $[k] = 2\pi \underline{r} [\underline{B}]^T [D][\underline{B}] \iint dr\, dz$

Or,

$$[k] \simeq [\underline{B}]^T [D][\underline{B}] 2\pi \underline{r} A \qquad (4.3.13)$$

4.3.2 Stiffness Matrix of a Quadrilateral Element

The strain-displacement relation for axisymmetric problem derived earlier (eq.(4.3.5)) can be rewritten as

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_r \\ \varepsilon_z \\ \varepsilon_\theta \\ \gamma_{rz} \end{Bmatrix} = \begin{Bmatrix} \dfrac{\partial u}{\partial r} \\ \dfrac{\partial w}{\partial z} \\ \dfrac{u}{r} \\ \dfrac{\partial u}{\partial z} + \dfrac{\partial w}{\partial r} \end{Bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & \dfrac{1}{r} \\ 0 & 1 & 1 & 0 & 0 \end{bmatrix} \begin{Bmatrix} \dfrac{\partial u}{\partial r} \\ \dfrac{\partial u}{\partial z} \\ \dfrac{\partial w}{\partial r} \\ \dfrac{\partial w}{\partial z} \\ u \end{Bmatrix} \qquad (4.3.14)$$

Applying chain rule of differentiation equation we get,

$$\begin{Bmatrix} \dfrac{\partial u}{\partial r} \\ \dfrac{\partial u}{\partial z} \\ \dfrac{\partial w}{\partial r} \\ \dfrac{\partial w}{\partial z} \\ u \end{Bmatrix} = \begin{bmatrix} J_{11}^* & J_{12}^* & 0 & 0 & 0 \\ J_{21}^* & J_{22}^* & 0 & 0 & 0 \\ 0 & 0 & J_{11}^* & J_{12}^* & 0 \\ 0 & 0 & J_{21}^* & J_{22}^* & 0 \\ 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} \dfrac{\partial u}{\partial \xi} \\ \dfrac{\partial u}{\partial \eta} \\ \dfrac{\partial w}{\partial \xi} \\ \dfrac{\partial w}{\partial \eta} \\ u \end{Bmatrix} \qquad (4.3.15)$$

Hence, the strain components are calculated as

$$\begin{Bmatrix} \varepsilon_r \\ \varepsilon_z \\ \varepsilon_\theta \\ \gamma_{rz} \end{Bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{r} \\ 0 & 1 & 1 & 0 & 0 \end{bmatrix} \begin{bmatrix} J_{11}^* & J_{12}^* & 0 & 0 & 0 \\ J_{21}^* & J_{22}^* & 0 & 0 & 0 \\ 0 & 0 & J_{11}^* & J_{12}^* & 0 \\ 0 & 0 & J_{21}^* & J_{22}^* & 0 \\ 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} \frac{\partial u}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \\ \frac{\partial w}{\partial \xi} \\ \frac{\partial w}{\partial \eta} \\ u \end{Bmatrix}$$

Or,

$$\begin{Bmatrix} \varepsilon_r \\ \varepsilon_z \\ \varepsilon_\theta \\ \gamma_{rz} \end{Bmatrix} = \begin{bmatrix} J_{11}^* & J_{12}^* & 0 & 0 & 0 \\ 0 & 0 & J_{21}^* & J_{22}^* & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{r} \\ J_{21}^* & J_{22}^* & J_{11}^* & J_{12}^* & 0 \end{bmatrix} \begin{Bmatrix} \frac{\partial u}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \\ \frac{\partial w}{\partial \xi} \\ \frac{\partial w}{\partial \eta} \\ u \end{Bmatrix} \qquad (4.3.16)$$

With the use of interpolation function and nodal displacements, $\left(\frac{\partial u}{\partial \xi}, \frac{\partial u}{\partial \eta}, \frac{\partial w}{\partial \xi}, \frac{\partial w}{\partial \eta}\right)$ can be expressed for a four node quadrilateral element as

$$\begin{Bmatrix} \frac{\partial u}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \\ \frac{\partial w}{\partial \xi} \\ \frac{\partial w}{\partial \eta} \end{Bmatrix} = \begin{bmatrix} \frac{\partial N_1}{\partial \xi} & \frac{\partial N_2}{\partial \xi} & \frac{\partial N_3}{\partial \xi} & \frac{\partial N_4}{\partial \xi} & 0 & 0 & 0 & 0 \\ \frac{\partial N_1}{\partial \eta} & \frac{\partial N_2}{\partial \eta} & \frac{\partial N_3}{\partial \eta} & \frac{\partial N_4}{\partial \eta} & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{\partial N_1}{\partial \xi} & \frac{\partial N_2}{\partial \xi} & \frac{\partial N_3}{\partial \xi} & \frac{\partial N_4}{\partial \xi} \\ 0 & 0 & 0 & 0 & \frac{\partial N_1}{\partial \eta} & \frac{\partial N_2}{\partial \eta} & \frac{\partial N_3}{\partial \eta} & \frac{\partial N_4}{\partial \eta} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ w_1 \\ w_2 \\ w_3 \\ w_4 \end{Bmatrix} \qquad (4.3.17)$$

Putting eq. (4.3.17) in eq. (4.3.16) we get,

$$\begin{Bmatrix} \varepsilon_r \\ \varepsilon_z \\ \varepsilon_\theta \\ \gamma_{rz} \end{Bmatrix} = \begin{bmatrix} J^*_{11} & J^*_{12} & 0 & 0 & 0 \\ 0 & 0 & J^*_{21} & J^*_{22} & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{r} \\ J^*_{21} & J^*_{22} & J^*_{11} & J^*_{12} & 0 \end{bmatrix} \begin{bmatrix} \frac{\partial N_1}{\partial \xi} & \frac{\partial N_2}{\partial \xi} & \frac{\partial N_3}{\partial \xi} & \frac{\partial N_4}{\partial \xi} & 0 & 0 & 0 & 0 \\ \frac{\partial N_1}{\partial \eta} & \frac{\partial N_2}{\partial \eta} & \frac{\partial N_3}{\partial \eta} & \frac{\partial N_4}{\partial \eta} & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{\partial N_1}{\partial \xi} & \frac{\partial N_2}{\partial \xi} & \frac{\partial N_3}{\partial \xi} & \frac{\partial N_4}{\partial \xi} \\ 0 & 0 & 0 & 0 & \frac{\partial N_1}{\partial \eta} & \frac{\partial N_2}{\partial \eta} & \frac{\partial N_3}{\partial \eta} & \frac{\partial N_4}{\partial \eta} \\ N_1 & N_2 & N_3 & N_4 & N_1 & N_2 & N_3 & N_4 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ w_1 \\ w_2 \\ w_3 \\ w_4 \end{Bmatrix} \tag{4.3.18}$$

Thus, the strain displacement relationship matrix [*B*] becomes

$$[B] = \begin{bmatrix} J^*_{11} & J^*_{12} & 0 & 0 & 0 \\ 0 & 0 & J^*_{21} & J^*_{22} & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{r} \\ J^*_{21} & J^*_{22} & J^*_{11} & J^*_{12} & 0 \end{bmatrix} \begin{bmatrix} \frac{\partial N_1}{\partial \xi} & \frac{\partial N_2}{\partial \xi} & \frac{\partial N_3}{\partial \xi} & \frac{\partial N_4}{\partial \xi} & 0 & 0 & 0 & 0 \\ \frac{\partial N_1}{\partial \eta} & \frac{\partial N_2}{\partial \eta} & \frac{\partial N_3}{\partial \eta} & \frac{\partial N_4}{\partial \eta} & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{\partial N_1}{\partial \xi} & \frac{\partial N_2}{\partial \xi} & \frac{\partial N_3}{\partial \xi} & \frac{\partial N_4}{\partial \xi} \\ 0 & 0 & 0 & 0 & \frac{\partial N_1}{\partial \eta} & \frac{\partial N_2}{\partial \eta} & \frac{\partial N_3}{\partial \eta} & \frac{\partial N_4}{\partial \eta} \\ N_1 & N_2 & N_3 & N_4 & N_1 & N_2 & N_3 & N_4 \end{bmatrix} \tag{4.3.19}$$

For a four node quadrilateral element,

$$\begin{aligned} N_1 &= \frac{(1-\xi)(1-\eta)}{4} \Rightarrow \frac{\partial N_1}{\partial \xi} = -\frac{(1-\eta)}{4} \text{ and } \frac{\partial N_1}{\partial \eta} = -\frac{(1-\xi)}{4} \\ N_2 &= \frac{(1+\xi)(1-\eta)}{4} \Rightarrow \frac{\partial N_2}{\partial \xi} = \frac{(1-\eta)}{4} \text{ and } \frac{\partial N_1}{\partial \eta} = -\frac{(1+\xi)}{4} \\ N_3 &= \frac{(1+\xi)(1+\eta)}{4} \Rightarrow \frac{\partial N_2}{\partial \xi} = \frac{(1+\eta)}{4} \text{ and } \frac{\partial N_1}{\partial \eta} = \frac{(1+\xi)}{4} \\ N_4 &= \frac{(1-\xi)(1+\eta)}{4} \Rightarrow \frac{\partial N_2}{\partial \xi} = -\frac{(1+\eta)}{4} \text{ and } \frac{\partial N_1}{\partial \eta} = \frac{(1-\xi)}{4} \end{aligned} \tag{4.3.20}$$

Thus, the [B] matrix will become

$$[\mathrm{B}]=\begin{bmatrix} J_{11}^{*} & J_{12}^{*} & 0 & 0 & 0 \\ 0 & 0 & J_{21}^{*} & J_{22}^{*} & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{r} \\ J_{21}^{*} & J_{22}^{*} & J_{11}^{*} & J_{12}^{*} & 0 \end{bmatrix}\times$$

$$\begin{bmatrix} -\frac{(1-\eta)}{4} & \frac{(1-\eta)}{4} & \frac{(1+\eta)}{4} & -\frac{(1+\eta)}{4} & 0 & 0 & 0 & 0 \\ -\frac{(1-\xi)}{4} & -\frac{(1+\xi)}{4} & \frac{(1+\xi)}{4} & \frac{(1-\xi)}{4} & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -\frac{(1-\eta)}{4} & \frac{(1-\eta)}{4} & \frac{(1+\eta)}{4} & -\frac{(1+\eta)}{4} \\ 0 & 0 & 0 & 0 & -\frac{(1-\xi)}{4} & -\frac{(1+\xi)}{4} & \frac{(1+\xi)}{4} & \frac{(1-\xi)}{4} \\ \frac{(1-\xi)(1-\eta)}{4} & \frac{(1+\xi)(1-\eta)}{4} & \frac{(1+\xi)(1+\eta)}{4} & \frac{(1-\xi)(1+\eta)}{4} & \frac{(1-\xi)(1-\eta)}{4} & \frac{(1+\xi)(1-\eta)}{4} & \frac{(1+\xi)(1+\eta)}{4} & \frac{(1-\xi)(1+\eta)}{4} \end{bmatrix} \tag{4.3.21}$$

The stiffness matrix for the axisymmetric element finally can be found from the following expression after numerical integration.

$$[k]=\int_{\Omega}[B]^{T}[D][B]\,d\Omega=\int_{-1}^{+1}\int_{-1}^{+1}[\mathrm{B}]^{\mathrm{T}}[\mathrm{D}][\mathrm{B}].2\pi\mathrm{r}.|J|.\mathrm{d}\xi\mathrm{d}\eta \tag{4.3.22}$$

SCHOOL OF MECHANICAL ENGINEERING
DEPARTMENT OF AERONAUTICAL ENGINEERING

UNIT – V –FINITE ELEMENT METHOD FOR AIRCRAFT STRUCTURES – SAEA1504

UNIT – V

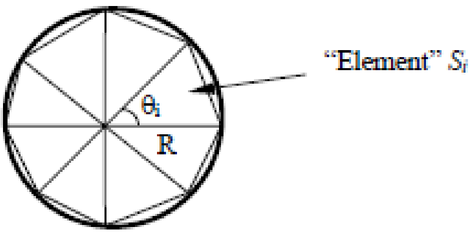
APPLICATION FINITE ELEMENT METHODS ELEMENT

I. Basic Concepts

The *finite element method* (FEM), or *finite element analysis* (FEA), is based on the idea of building a complicated object with simple blocks, or, dividing a complicated object into small and manageable pieces. Application of this simple idea can be found everywhere in everyday life, as well as in engineering.

Examples:

- Lego (kids' play)
- Buildings
- Approximation of the area of a circle:



Area of one triangle: $S_i = \frac{1}{2} R^2 \sin\theta_i$

Area of the circle: $S_N = \sum_{i=1}^{N} S_i = \frac{1}{2} R^2 N \sin\left(\frac{2\pi}{N}\right) \rightarrow \pi R^2 \ as\ N \rightarrow \infty$

where N = total number of triangles (elements).

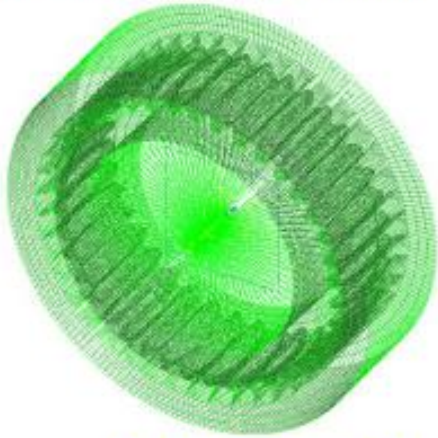
Observation: Complicated or smooth objects can be represented by geometrically simple pieces (elements).

Why Finite Element Method?

- *Design analysis*: hand calculations, experiments, and computer simulations
- FEM/FEA is the most widely applied computer simulation method in engineering
- Closely integrated with CAD/CAM applications
- ...

Applications of FEM in Engineering

- Mechanical/Aerospace/Civil/Automobile Engineering
- Structure analysis (static/dynamic, linear/nonlinear)
- Thermal/fluid flows
- Electromagnetics
- Geomechanics
- Biomechanics
- ...



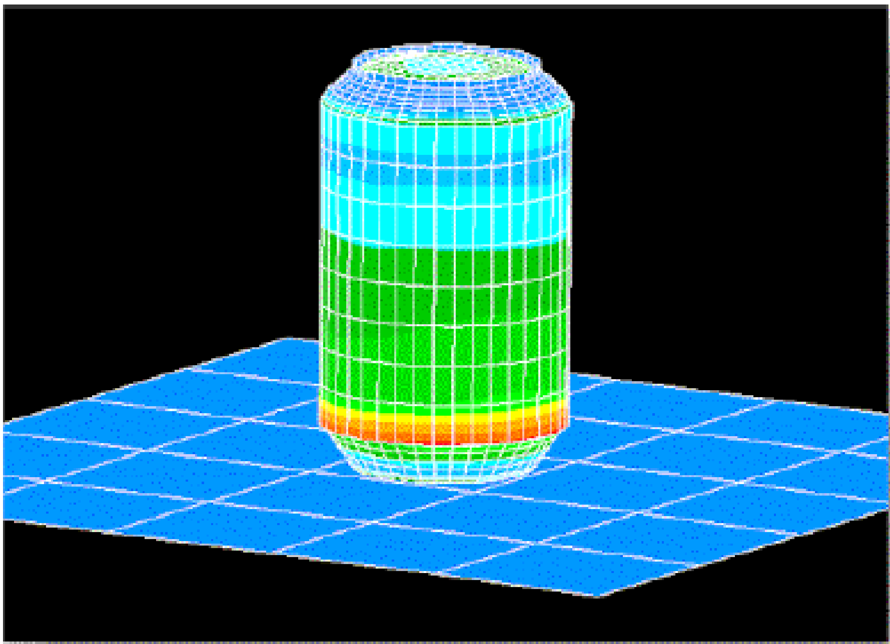
Modeling of gear coupling

Examples:

A Brief History of the FEM

- 1943 ----- Courant (Variational methods)
- 1956 ----- Turner, Clough, Martin and Topp (Stiffness)
- 1960 ----- Clough ("Finite Element", plane problems)

- 1970s ----- Applications on mainframe computers
- 1980s ----- Microcomputers, pre- and postprocessors
- 1990s ----- Analysis of large structural systems

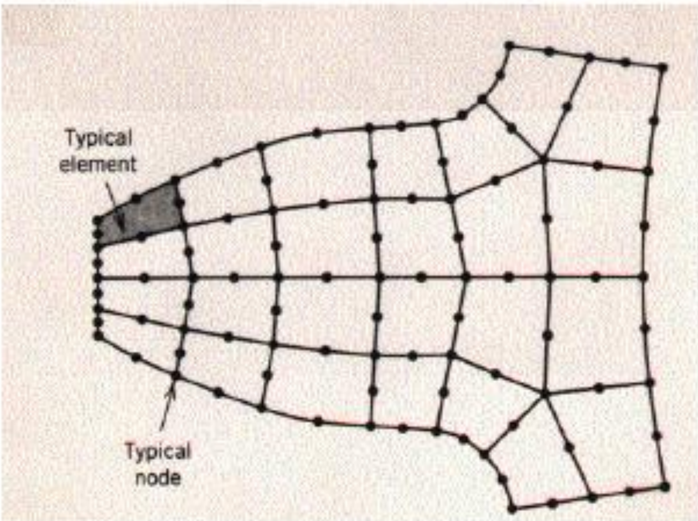


Can Drop Test (Click for more information and an animation)

FEM in Structural Analysis (The Procedure)

- Divide structure into pieces (elements with nodes)
- Describe the behavior of the physical quantities on each element
- Connect (assemble) the elements at the nodes to form an approximate system of equations for the whole structure
- Solve the system of equations involving unknown quantities at the nodes (e.g., displacements)
- Calculate desired quantities (e.g., strains and stresses) at selected elements

Example:



FEM model for a gear tooth (From Cook's book, p.2).

Computer Implementations

- Preprocessing (build FE model, loads and constraints)
- FEA solver (assemble and solve the system of equations)
- Postprocessing (sort and display the results)

Available Commercial FEM Software Packages

- *ANSYS* (General purpose, PC and workstations)
- *SDRC/I-DEAS* (Complete CAD/CAM/CAE package)
- *NASTRAN* (General purpose FEA on mainframes)
- *ABAQUS* (Nonlinear and dynamic analyses)
- *COSMOS* (General purpose FEA)
- *ALGOR* (PC and workstations)
- *PATRAN* (Pre/Post Processor)
- *HyperMesh* (Pre/Post Processor)
- *Dyna-3D* (Crash/impact analysis)
- ...

A Link to CAE Software and Companies

II. Substructures (Superelements)

Substructuring is a process of analyzing a large structure as a collection of (natural) components. The FE models for these components are called *substructures* or *superelements* (SE).

Physical Meaning:

A finite element model of a portion of structure.

Mathematical Meaning:

Boundary matrices which are load and stiffness matrices reduced (condensed) from the *interior* points to the *exterior* or boundary points.

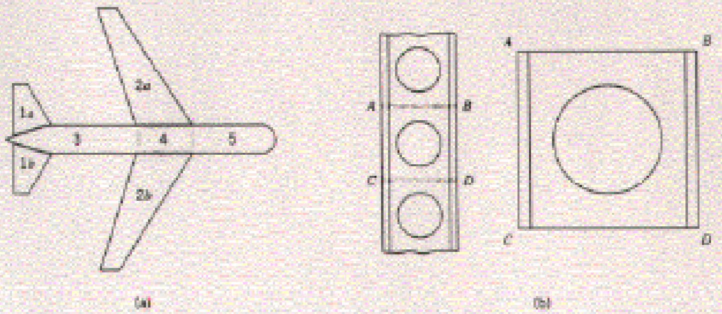


Fig. 4.11-1. (a) Possible substructures 1a, 1b, ..., 5 of a hypothetical aircraft. (b) Castellated beam, with typical repeating substructure *ABCD*. Elements of the substructures are not shown.

Advantages of Using Substructures/Superelements:

- Large problems (which will otherwise exceed your computer capabilities)
- Less CPU time per run once the superelements have been processed (i.e., matrices have been saved)
- Components may be modeled by different groups
- Partial redesign requires only partial reanalysis (reduced cost)
- Efficient for problems with local nonlinearities (such as confined plastic deformations) which can be placed in one superelement (residual structure)
- Exact for static stress analysis

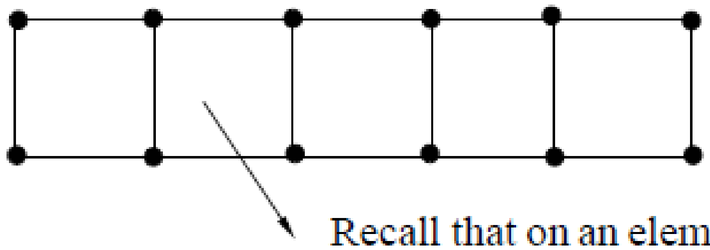
Disadvantages:

- Increased overhead for file management
- Matrix condensation for dynamic problems introduce new approximations
- ...

IV. Nature of Finite Element Solutions

- FE Model – A mathematical model of the real structure, based on many approximations.
- Real Structure -- Infinite number of nodes (physical points or particles), thus infinite number of DOF's.
- FE Model – finite number of nodes, thus finite number of DOF's.

⇨ Displacement field is controlled (or constrained) by the values at a limited number of nodes.



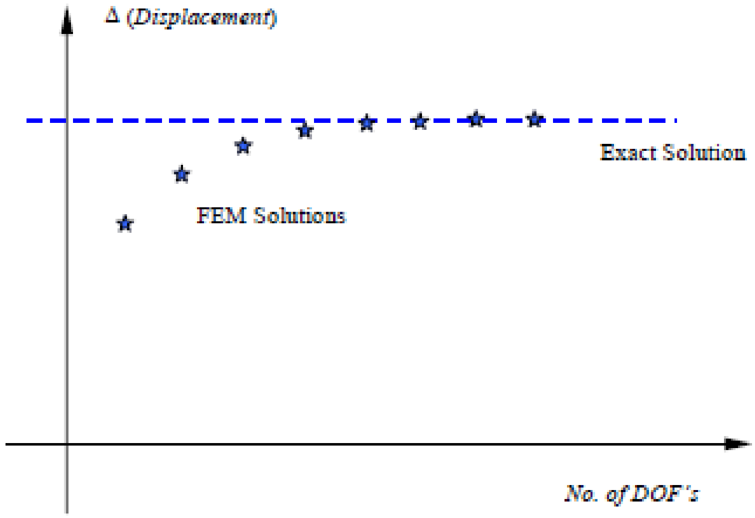
Recall that on an element :

$$u = \sum_{\alpha=1}^{4} N_\alpha u_\alpha$$

Stiffening Effect:

- FE Model is stiffer than the real structure.
- In general, displacement results are smaller in magnitudes than the exact values.

Hence, FEM solution of displacement provides a *lower bound* of the exact solution.



The FEM solution approaches the exact solution from below.

This is true for displacement based FEA!

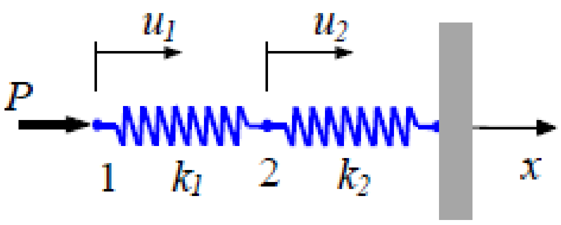
V. Numerical Error

Error $\neq$ Mistakes in FEM (modeling or solution).

Type of Errors:

- Modeling Error (beam, plate … theories)
- Discretization Error (finite, piecewise …)
- Numerical Error (in solving FE equations)

Example (numerical error):

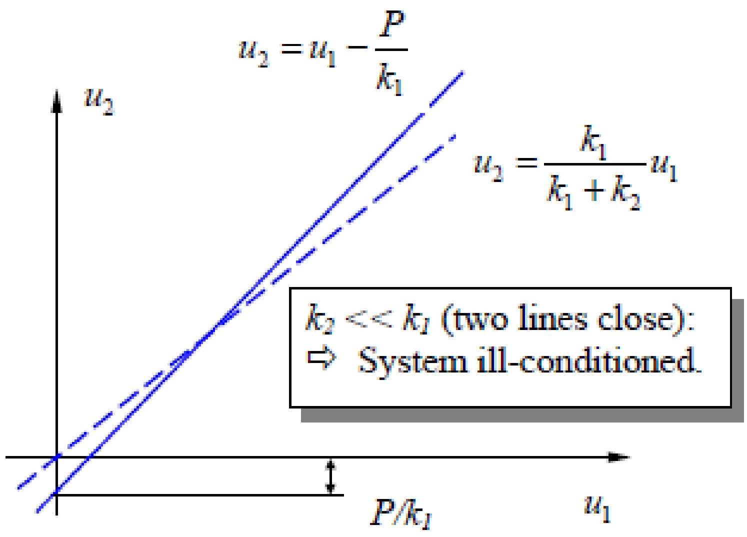


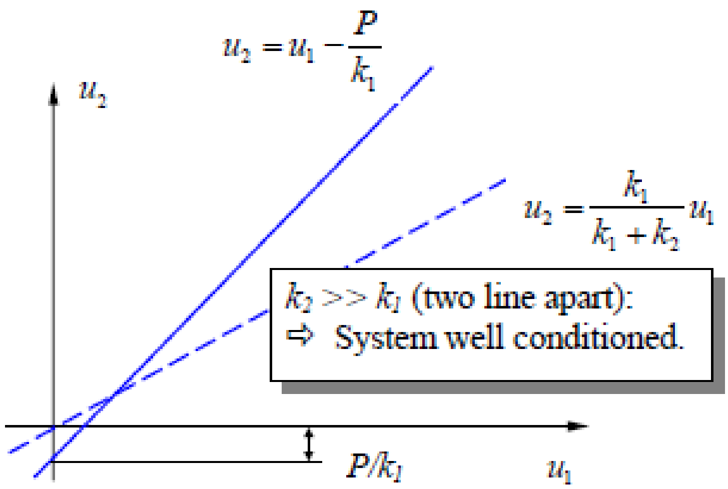
FE Equations:

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 + k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} P \\ 0 \end{Bmatrix}$$

and $\qquad Det\,\mathbf{K} = k_1 k_2$.

The system will be *singular* if k_2 is small compared with k_1.





- Large difference in stiffness of different parts in FE model may cause ill-conditioning in FE equations. Hence giving results with large errors.
- Ill-conditioned system of equations can lead to large changes in solution with small changes in input (right hand side vector).

VI. Convergence of FE Solutions

As the mesh in an FE model is "refined" repeatedly, the FE solution will converge to the exact solution of the mathematical model of the problem (the model based on bar, beam, plane stress/strain, plate, shell, or 3-D elasticity theories or assumptions).

Type of Refinements:

h-refinement:	reduce the size of the element ("*h*" refers to the typical size of the elements);
p-refinement:	Increase the order of the polynomials on an element (linear to quadratic, etc.; "*h*" refers to the highest order in a polynomial);
r-refinement:	re-arrange the nodes in the mesh;
hp-refinement:	Combination of the h- and p-refinements (better results!).

Examples:

…

VII. Adaptivity (h-, p-, and hp-Methods)

- Future of FE applications
- Automatic refinement of FE meshes until converged results are obtained
- User's responsibility reduced: only need to generate a good initial mesh

Error Indicators:

Define,

σ --- element by element stress field (discontinuous),

σ^*--- averaged or smooth stress (continuous),

$\sigma_E = \sigma - \sigma^*$ --- the error stress field.

Compute strain energy,

$$U = \sum_{i=1}^{M} U_i, \qquad U_i = \int_{V_i} \frac{1}{2} \boldsymbol{\sigma}^T \mathbf{E}^{-1} \boldsymbol{\sigma} dV;$$

$$U^* = \sum_{i=1}^{M} U_i^*, \qquad U_i^* = \int_{V_i} \frac{1}{2} \boldsymbol{\sigma}^{*T} \mathbf{E}^{-1} \boldsymbol{\sigma}^* dV;$$

$$U_E = \sum_{i=1}^{M} U_{Ei}, \qquad U_{Ei} = \int_{V_i} \frac{1}{2} \boldsymbol{\sigma}_E^T \mathbf{E}^{-1} \boldsymbol{\sigma}_E dV;$$

where M is the total number of elements, V_i is the volume of the element i.

One error indicator --- the relative energy error:

$$\eta = \left[\frac{U_E}{U + U_E} \right]^{1/2} . \qquad (0 \le \eta \le 1)$$

The indicator η is computed after each FE solution. Refinement of the FE model continues until, say

$\eta \le 0.05$.

=> converged FE solution.

Interpolation Functions for General Element Formulation

In finite element analysis, solution accuracy is judged in terms of convergence as the element "mesh" is refined.
There are two major methods of mesh refinement.

In the first, known as ***h-refinement***, mesh refinement refers to the process of increasing the number of elements used to model a given domain, consequently, reducing individual element size.

In the second method, ***p-refinement***, element size is unchanged but the order of the polynomials used as interpolation functions is increased.

The objective of mesh refinement in either method is to obtain sequential solutions that exhibit asymptotic convergence to values representing the exact solution.

TWO DIMENSIONAL STEADY STATE HEAT FLOW:

Introduction

In finite element technique, the nodal equations for the field variables are obtained through an integral formulation, which may be set up through a variational principle (if one exists), or through the Galekin's weighted residual approach. Here we shall consider the Galerkin's approach, which has a general applicability. Let us consider a general representation of a differential equation on a region V

$$LT = Q \tag{18.1}$$

For the one dimensional heat conduction equation, the governing differential equation is

$$\frac{d}{dx}\left(kA\frac{dT}{dx}\right) = 0 \tag{18.2}$$

The symbol L is an operator

$$\frac{d}{dx}kA\frac{d}{dx}$$

that is operating on T. The exact solution requires to satisfy Eqn. 18.1 at every x points. Let us seek for an approximate solution $\bar{T}$ that introduces an error $\varepsilon(x)$, called the **residual**

$$\varepsilon(x) = L\bar{T} - Q \tag{18.3}$$

The approximate methods are centered around the concept of setting the residual relative to a weighting function W_i to zero

$$\int_V W_i\left(L\bar{T} - Q\right) dV = 0 \quad i - 1 \; to \; n \tag{18.4}$$

The W_i can be chosen based on the guiding philosophies of different variants of the weighted residual methods. In the Galerkin method, the W_i are chosen from the **basis functions** used for constructing $\bar{T}$. We shall deal with aspect, in detail, in the subsequent sections.

Formulation

Consider a steady 2-D heat conduction problem in an arbitrary-shaped two dimensional domain which is subject to various types of boundary conditions as shown in Fig. 18.1. Considering a uniform heat generation rate per unit volume (Q) in the entire solution, the governing equation for heat transfer is

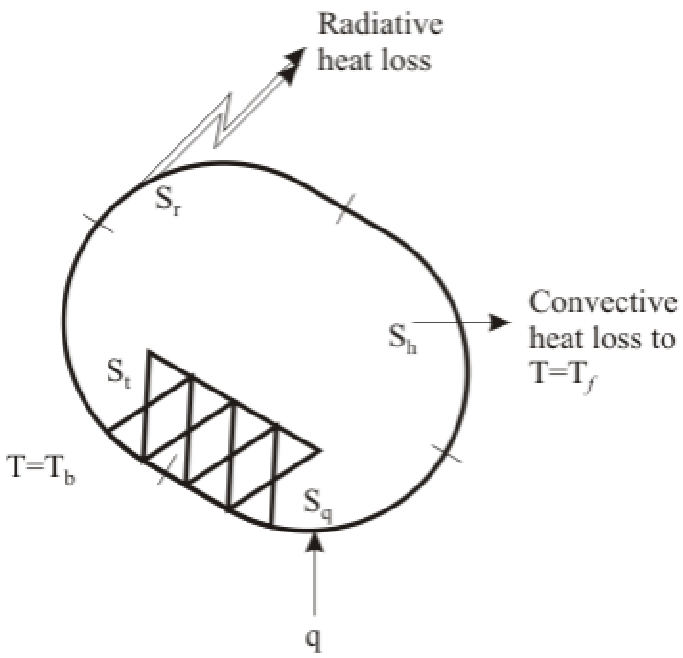


Figure 18.1 Arbitrary 2D domain.

$$k\nabla^2 T + Q = 0 \tag{18.5}$$

where T(x,y) is the exact distribution of temperature.

The boundary conditions are:

$$T = T_b \text{ on } S_t$$
$$-k\frac{\partial T}{\partial n} = q \text{ on } S_q$$
$$-k\frac{\partial T}{\partial n} = h\left(T - T_f\right) \text{ on } S_h \text{ and}$$
$$-k\frac{\partial T}{\partial n} = \sigma\varepsilon_s\left(T^4 - T_f^4\right) \text{ on } S_r \tag{18.6}$$

TWO DIMENSIONAL STEADY STATE HEAT FLOW:

We have, to this point, considered only One Dimensional, Steady State problems. The reason for this is that such problems lead to ordinary differential equations and can be solved with relatively ordinary mathematical techniques. In general the properties of any physical system may depend on both location (x, y, z) and time (τ). The inclusion of two or more independent variables results in a partial differential equation. The multidimensional heat diffusion equation in a Cartesian coordinate system can be written as:

$$\frac{1}{a}\cdot\frac{\partial T}{\partial \tau}=\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}+\frac{\hat{q}}{k} \qquad (1)$$

The above equation governs the Cartesian, temperature distribution for a three-dimensional unsteady, heat transfer problem involving heat generation. To solve for the full equation, it requires a total of six boundary conditions: two for each direction. Only one initial condition is needed to account for the transient behavior. For 2D, steady state (∂/ ∂t = 0) and without heat generation, the above equation reduces to:

$$\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}=0 \qquad (2)$$

Equation (2) needs 2 boundary conditions in each direction. There are three approaches to solve this equation:

- Analytical Method: The mathematical equation can be solved using techniques like the method of separation of variables.
- Graphical Method: Limited use. However, the conduction shape factor concept derived under this concept can be useful for specific configurations. (see Table 4.1 for selected configurations)
- Numerical Method: Finite difference or finite volume schemes, usually will be solved using computers.

Analytical solutions are possible only for a limited number of cases (such as linear problems with simple geometry). Standard analytical techniques such as separation of variables can be found in basic textbooks on engineering mathematics, and will not be reproduced here. The student is encouraged to refer to textbooks on basic mathematics for an overview of the analytical solutions to heat diffusion problems. In the present lecture material, we will cover the graphical and numerical techniques, which are used quite conveniently by engineers for solving multi-dimensional heat conduction problems.

Graphical Method: Conduction Shape Factor

This approach applied to 2-D conduction involving two isothermal surfaces, with all other surfaces being adiabatic. The heat transfer from one surface (at a temperature T1) to the other surface (at T2) can be expressed as: q=Sk(T1-T2) where k is the thermal conductivity of the solid and S is the conduction shape factor.

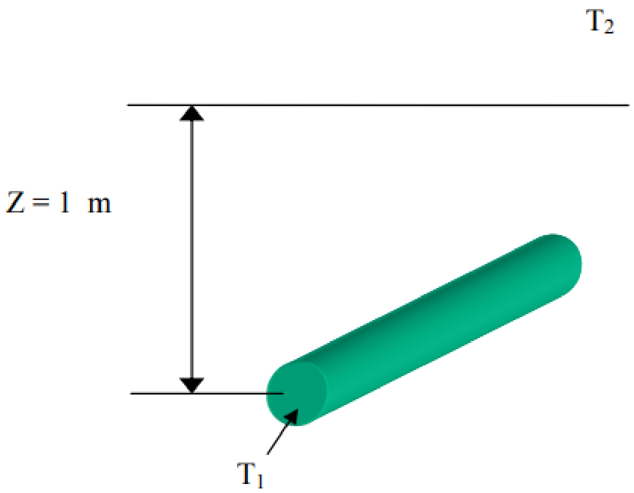
The shape factor can be related to the thermal resistance:

$$q=S.k.(T_1-T_2)=(T_1-T_2)/(1/kS)= (T_1-T_2)/R_t$$

where Rt = 1/(kS) is the thermal resistance in 2D. Note that 1-D heat transfer can also use the concept of shape factor. For example, heat transfer inside a plane wall of thickness L is q=kA(ΔT/L), where the shape factor S=A/L. Common shape factors for selected configurations can be found in Table 4.1

Example: A 10 cm OD uninsulated pipe carries steam from the power plant across campus. Find the heat loss if the pipe is buried 1 m in the ground is the ground surface temperature is 50 ºC. Assume a thermal conductivity of the sandy soil as k = 0.52 w/m K.

Solution:



The shape factor for long cylinders is found in Table 4.1 as Case 2, with L >> D:

$$S = 2 \cdot \pi \cdot L/\ln(4 \cdot z/D)$$

Where z = depth at which pipe is buried.

$$S = 2 \cdot \pi \cdot 1 \cdot m/\ln(40) = 1.7 \text{ m}$$

Then

$$q' = (1.7 \cdot m)(0.52 \text{ W/m} \cdot \text{K})(100\,^{o}\text{C} - 50\,^{o}\text{C})$$

$$q' = 44.2 \text{ W}$$

Table 4.1

Conduction shape factors for selected two-dimensional systems [$q = Sk(T_1 - T_2)$]

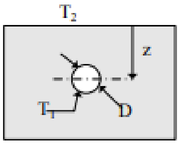
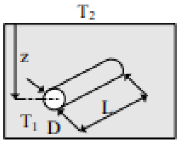
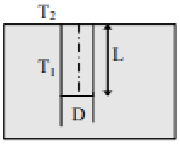
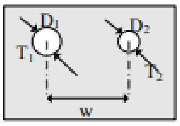
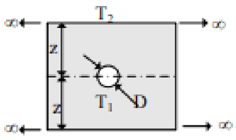
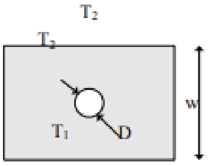
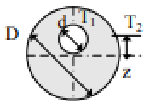
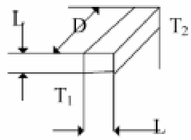
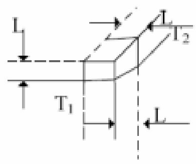
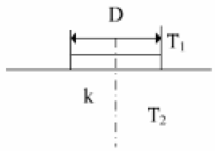
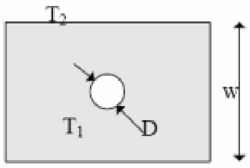
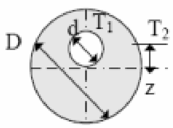
System	Schematic	Restrictions	Shape Factor
Isothermal sphere buried in as finite medium		$z > D/2$	$\frac{2\pi D}{1 - D/4z}$
Horizontal isothermal cylinder of length L buried in a semi finite medium		$L \gg D$ $L \gg D$, $z > 3D/2$	$\frac{2\pi L}{\cosh^{-1}(2z/D)}$ $\frac{2\pi L}{\ln(4z/D)}$
Vertical cylinder in a semi finite medium		$L \gg D$	$\frac{2\pi L}{\ln(4L/D)}$
Conduction between two cylinders of length L in infinite medium		$L \gg D_1, D_2$ $L \gg w$	$\frac{2\pi L}{\cosh^{-1}\left(\frac{4w^2 - D_1^2 - D_2^2}{2D_1 D_2}\right)}$
Horizontal circular cylinder of length L midway between parallel planes of equal length and infinite width		$z \gg D/2$ $L \gg 2$	$\frac{2\pi L}{\ln(8z/\pi D)}$
Circular cylinder of length L centered in a square solid of equal length		$W > D$ $L \gg w$	$\frac{2\pi L}{\ln(1.08w/D)}$
Eccentric circular cylinder of length L in a cylinder of equal length		$D > d$ $L \gg D$	$\frac{2\pi L}{\cosh^{-1}\left(\frac{D^2 + d^2 - 4z^2}{2Dd}\right)}$

Table 4.1 Continued

System	Schematic	Restrictions	Shape Factor
Conduction through the edge of adjoining walls		D>L/5	0.54D
Conduction through corners of three walls with a temperature difference of ΔT_{1-2} across the walls		L<<length and width of wall	0.15L
Disk of diameter D and T1 on a semi finite medium of thermal conductivity k and T_2		None	2D
Circular cylinder of length L centered in a square solid of equal length		W>D L>>w	$\frac{2\pi L}{\ln(1.08w/D)}$
Eccentric circular cylinder of length L in a cylinder of equal length		D>d L>>D	$\frac{2\pi L}{\cosh^{-1}\left(\frac{D^2+d^2-4z^2}{2Dd}\right)}$

Due to the increasing complexities encountered in the development of modern technology, analytical solutions usually are not available. For these problems, numerical solutions obtained using high-speed computer are very useful, especially when the geometry of the object of interest is irregular, or the boundary conditions are nonlinear. In numerical analysis, three different approaches are commonly used: the finite difference, the finite volume and the finite element methods. Brief descriptions of the three methods are as follows

The Finite Difference Method (FDM) This is the oldest method for numerical solution of PDEs, introduced by Euler in the 18th century. It's also the easiest method to use for simple geometries. The starting point is the conservation equation in differential form. The solution domain is covered by grid. At each grid point, the differential equation is approximated by replacing the partial derivatives by approximations in terms of the nodal values of the functions. The result is one algebraic equation per grid node, in which the variable value at that and a certain number of neighbor nodes appear as unknowns. In principle, the FD method can be applied to any grid type. However, in all applications of the FD method known, it has been applied to structured grids. Taylor series expansion or polynomial fitting is used to obtain approximations to the first and second derivatives of the variables with respect to the coordinates. When necessary, these methods are also used to obtain variable values at locations other than grid nodes (interpolation). On structured grids, the FD method is very simple and effective. It is especially easy to obtain higher-order schemes on regular grids. The disadvantage of FD methods is that the conservation is not enforced unless special care is taken. Also, the restriction to simple geometries is a significant disadvantage.

Finite Volume Method (FVM)

In this dissertation finite volume method is used. The FV method uses the integral form of the conservation equations as its starting point. The solution domain is subdivided into a finite number of contiguous control volumes (CVs), and the conservation equations are applied to each CV. At the centroid of each CV lies a computational node at which the variable values are to be calculated. Interpolation is used to express variable values at the CV surface in terms of the nodal (CV-center) values. As a result, one obtains an algebraic equation for each CV, in which a number of neighbor nodal values appear. The FVM method can accommodate any type of grid when compared to FDM, which is applied to only structured grids. The FVM approach is perhaps the simplest to understand and to program. All terms that need be approximated have physical meaning, which is why it is popular. The disadvantage of FV methods compared to FD schemes is that methods of order higher than second are more difficult to develop in 3D. This is due to the fact that the FV approach requires two levels of approximation: interpolation and integration.

Finite Element Method (FEM)

The FE method is similar to the FV method in many ways. The domain is broken into a set of discrete volumes or finite elements that are generally unstructured; in 2D, they are usually triangles or quadrilaterals, while in 3D tetrahedra or hexahedra are most often used. The distinguishing feature of FE methods is that the equations are multiplied by a weight function before they are integrated over the entire domain. In the simplest FE methods, the solution is approximated by a linear shape function within each element in a way that guarantees continuity of the solution across element boundaries. Such a function can be constructed from its values at the corners of the elements. The weight function is usually of the same form.

This approximation is then substituted into the weighted integral of the conservation law and the equations to be solved are derived by requiring the derivative of the integral with respect to each nodal value to be zero; this corresponds to selecting the best solution within the set of allowed functions (the one with minimum residual). The result is a set of non-linear algebraic equations.

An important advantage of finite element methods is the ability to deal with arbitrary geometries. Finite element methods are relatively easy to analyze mathematically and can be shown to have optimality properties for certain types of equations. The principal drawback, which is shared by any method that uses unstructured grids, is that the matrices of the linearized equations are not as well structured as those for regular grids making it more difficult to find efficient solution methods.

The Finite Difference Method Applied to Heat Transfer Problems:

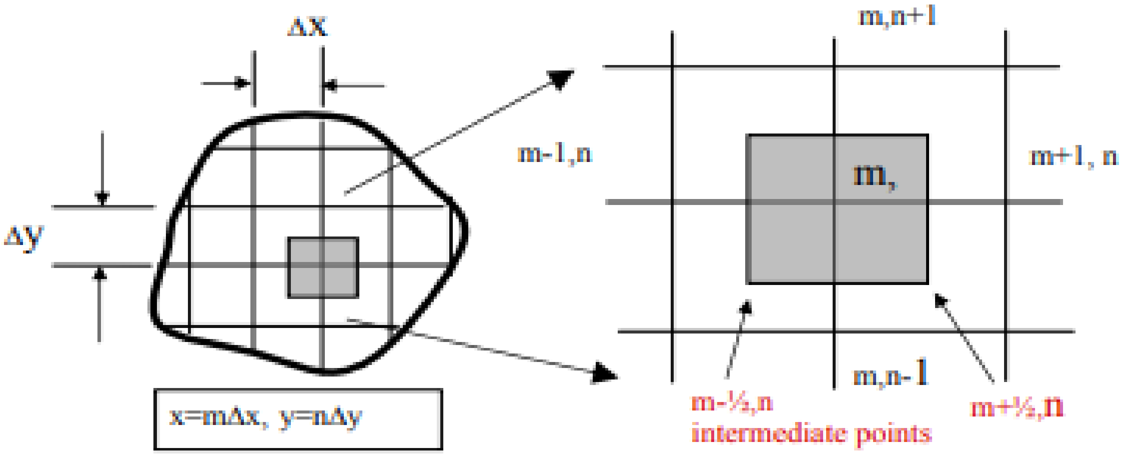
In heat transfer problems, the finite difference method is used more often and will be discussed here in more detail. The finite difference method involves:

* Establish nodal networks
* Derive finite difference approximations for the governing equation at both interior and exterior nodal points
* Develop a system of simultaneous algebraic nodal equations
* Solve the system of equations using numerical schemes

The Nodal Networks:

The basic idea is to subdivide the area of interest into sub-volumes with the distance between adjacent nodes by Δx and Δy as shown. If the distance between points is small enough, the differential equation can be approximated locally by a set of finite difference equations. Each node now represents a small region where the nodal temperature is a measure of the average temperature of the region.

Example:



Finite Difference Approximation:

Heat Diffusion Equation: $\nabla^2 T + \frac{\dot{q}}{k} = \frac{1}{\alpha}\frac{\partial T}{\partial t}$,

where $\alpha = \frac{k}{\rho C_p V}$ is the thermal diffusivity

No generation and steady state: $\dot{q}=0$ and $\frac{\partial}{\partial t} = 0, \Rightarrow \nabla^2 T = 0$

First, approximated the first order differentiation at intermediate points (m+1/2,n) & (m-1/2,n)

$$\left.\frac{\partial T}{\partial x}\right|_{(m+1/2,n)} \approx \left.\frac{\Delta T}{\Delta x}\right|_{(m+1/2,n)} = \frac{T_{m+1,n} - T_{m,n}}{\Delta x}$$

$$\left.\frac{\partial T}{\partial x}\right|_{(m-1/2,n)} \approx \left.\frac{\Delta T}{\Delta x}\right|_{(m-1/2,n)} = \frac{T_{m,n} - T_{m-1,n}}{\Delta x}$$

Next, approximate the second order differentiation at m,n

$$\left.\frac{\partial^2 T}{\partial x^2}\right|_{m,n} \approx \frac{\partial T/\partial x|_{m+1/2,n} - \partial T/\partial x|_{m-1/2,n}}{\Delta x}$$

$$\left.\frac{\partial^2 T}{\partial x^2}\right|_{m,n} \approx \frac{T_{m+1,n} + T_{m-1,n} - 2T_{m,n}}{(\Delta x)^2}$$

Similarly, the approximation can be applied to the other dimension y

$$\left.\frac{\partial^2 T}{\partial y^2}\right|_{m,n} \approx \frac{T_{m,n+1} + T_{m,n-1} - 2T_{m,n}}{(\Delta y)^2}$$

$$\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right)_{m,n} \approx \frac{T_{m+1,n}+T_{m-1,n}-2T_{m,n}}{(\Delta x)^2}+\frac{T_{m,n+1}+T_{m,n-1}-2T_{m,n}}{(\Delta y)^2}$$

To model the steady state, no generation heat equation: $\nabla^2 T = 0$

This approximation can be simplified by specify $\Delta x = \Delta y$

and the nodal equation can be obtained as

$$T_{m+1,n}+T_{m-1,n}+T_{m,n+1}+T_{m,n-1}-4T_{m,n}=0$$

This equation approximates the nodal temperature distribution based on the heat equation. This approximation is improved when the distance between the adjacent nodal points is decreased:

Since $\lim(\Delta x \to 0)\frac{\Delta T}{\Delta x}=\frac{\partial T}{\partial x}, \lim(\Delta y \to 0)\frac{\Delta T}{\Delta y}=\frac{\partial T}{\partial y}$

Table 4.2 provides a list of nodal finite difference equation for various configurations.

<u>A System of Algebraic Equations</u>

- The nodal equations derived previously are valid for all interior points satisfying the steady state, no generation heat equation. For each node, there is one such equation.

  For example: for nodal point m=3, n=4, the equation is

  $T_{2,4} + T_{4,4} + T_{3,3} + T_{3,5} - 4T_{3,4} = 0$

  $T_{3,4} = (1/4)(T_{2,4} + T_{4,4} + T_{3,3} + T_{3,5})$
- Nodal relation table for exterior nodes (boundary conditions) can be found in standard heat transfer textbooks.
- Derive one equation for each nodal point (including both interior and exterior points) in the system of interest. The result is a system of N algebraic equations for a total of N nodal points.

<u>Matrix Form</u>

The system of equations:

$$\begin{array}{l} a_{11}T_1 + a_{12}T_2 + \cdots + a_{1N}T_N = C_1 \\ a_{21}T_1 + a_{22}T_2 + \cdots + a_{2N}T_N = C_2 \\ \vdots \quad \vdots \quad \vdots \quad \vdots \quad \vdots \\ a_{N1}T_1 + a_{N2}T_2 + \cdots + a_{NN}T_N = C_N \end{array}$$

A total of N algebraic equations for the N nodal points and the system can be expressed as a matrix formulation: **[A][T]=[C]** .

$$\text{where } A=\begin{bmatrix} a_{11} & a_{12} & \cdots & a_{1N} \\ a_{21} & a_{22} & \cdots & a_{2N} \\ \vdots & \vdots & \vdots & \vdots \\ a_{N1} & a_{N2} & \cdots & a_{NN} \end{bmatrix}, T=\begin{bmatrix} T_1 \\ T_2 \\ \vdots \\ T_N \end{bmatrix}, C=\begin{bmatrix} C_1 \\ C_2 \\ \vdots \\ C_N \end{bmatrix}$$

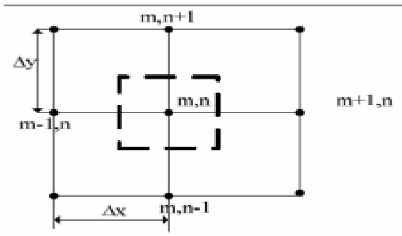
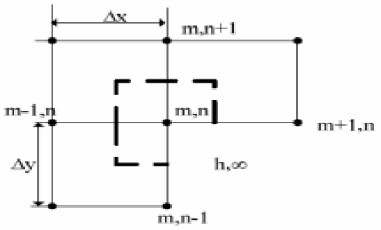
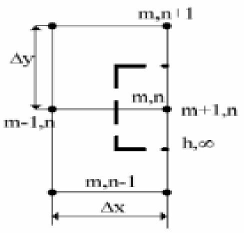
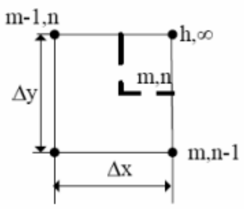
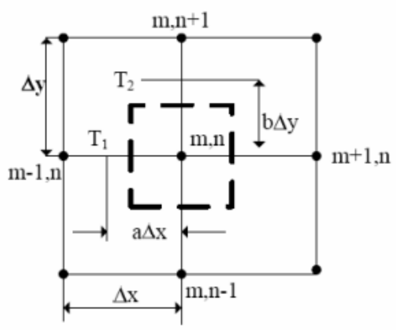
Table 4.2 Summary of nodal finite-difference methods	
Configuration	Finite-Difference equations for $\Delta x = \Delta y$
	$T_{m,n+1} + T_{m,n-1} + T_{m+1,n} + T_{m-1,n} - 4T_{m,n} = 0$ Case 1. Interior node
	$2(T_{m-1,n} + T_{m,n+1}) + (T_{m+1,n} + T_{m,n-1}) + 2\frac{h\Delta x}{k}T_\infty - 2\left(3 + \frac{h\Delta x}{k}\right)T_{m,n} = 0$ Case 2. Node at an internal corner with convection
	$2(T_{m-1,n} + T_{m,n+1} + T_{m,n-1}) + 2\frac{h\Delta x}{k}T_\infty - 2\left(\frac{h\Delta x}{k} + 2\right)T_{m,n} = 0$ Case 3. Node at a plane surface with convection

Table 4.2 Summary of nodal finite-difference methods	
	$2(T_{m-1,n} + T_{m,n+1} + T_{m,n-1}) + 2\frac{h\Delta x}{k}T_\infty - 2\left(\frac{h\Delta x}{k} + 1\right)T_{m,n} = 0$ Case 4. Node at an external corner with convection
	$\frac{2}{a+1}T_{m+1,n} + \frac{2}{b+1}T_{m,n-1} + \frac{2}{a(a+1)}T_1 + \frac{2}{b)b+1)}T_2 - \left(\frac{2}{a} + \frac{2}{b}\right)T_{m,n} = 0$ Case 5. Node near a curved surface maintained at a non uniform temperature

Numerical Solutions

Matrix form: [**A**][**T**]=[**C**].

From linear algebra: $[\mathbf{A}]^{-1}[\mathbf{A}][\mathbf{T}]=[\mathbf{A}]^{-1}[\mathbf{C}]$, $[\mathbf{T}]=[\mathbf{A}]^{-1}[\mathbf{C}]$

where [**A**]-1 is the inverse of matrix [**A**]. [**T**] is the solution vector.

- Matrix inversion requires cumbersome numerical computations and is not efficient if the order of the matrix is high (>10)
- Gauss elimination method and other matrix solvers are usually available in many numerical solution package. For example, "Numerical Recipes" by Cambridge University Press or their web source at www.nr.com.
- For high order matrix, iterative methods are usually more efficient. The famous Jacobi & Gauss-Seidel iteration methods will be introduced in the following.

Iteration

General algebraic equation for nodal point:

$$\sum_{j=1}^{i-1} a_{ij}T_j + a_{ii}T_i + \sum_{j=i+1}^{N} a_{ij}T_j = C_i,$$

(Example : $a_{31}T_1 + a_{32}T_2 + a_{33}T_3 + \cdots + a_{3N}T_N = C_i,\ i=3$)

Rewrite the equation of the form:

$$T_i^{(k)} = \frac{C_i}{a_{ii}} - \sum_{j=1}^{i-1}\frac{a_{ij}}{a_{ii}}T_j^{(k)} - \sum_{j=i+1}^{N}\frac{a_{ij}}{a_{ii}}T_j^{(k-1)}$$

Replace (k) by (k-1) for the Jacobi iteration

- (k) - specify the level of the iteration, (k-1) means the present level and (k) represents the new level.
- An initial guess (k=0) is needed to start the iteration.
- • By substituting iterated values at (k-1) into the equation, the new values at iteration (k) can be estimated The iteration will be stopped when max$|T_i^{(k)}-T_i^{(k-1)}|$ε≤ , where ε specifies a predetermined value of acceptable error