Finite Element Method

Module 1

Fundamental Concepts

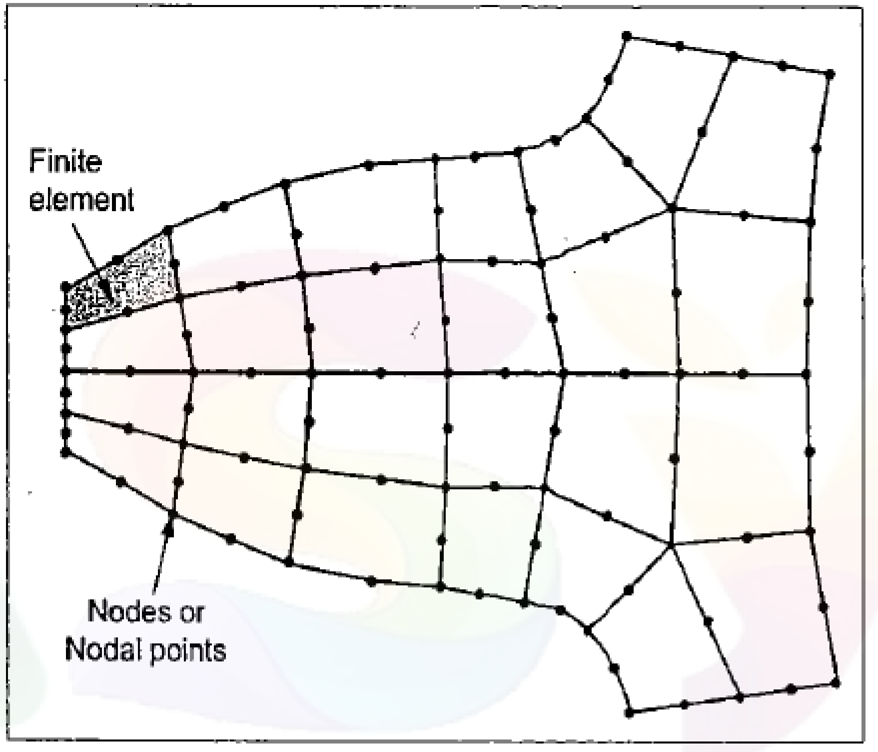
- Methods of Engineering Analysis
  - Experimental Methods
  - Analytical Methods/ Theoretical Analysis
  - Numerical Methods
    - – Functional Approximation (weighted Residual or Galerkin method and Rayleigh Ritz method or variational approach)
    - – Finite Differential Methods
    - – Finite Element Methods/ Finite Element Analysis

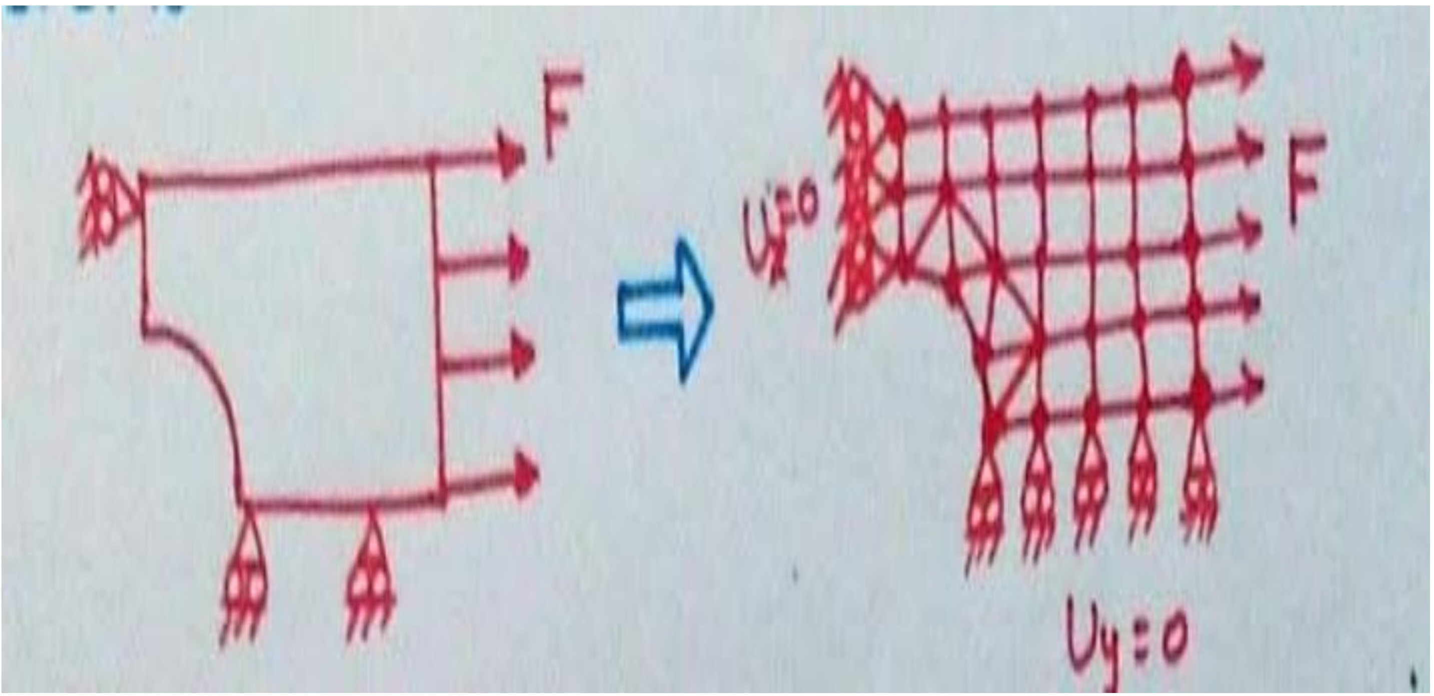
Finite Element Method

- The finite element method is to find the solution of a complicated problem by replacing it by a simpler one. Here the actual problem is replaced by a simpler one in finding the solution, we will be able to find only an approximate solution rather than the exact solution.
- The existing mathematical tools will not be sufficient to find the exact solution (and sometimes, even an approximate solution) of most of the practical problems. Thus, in the absence of any other convenient method to find even the approximate solution of a given problem, we have to prefer the finite element method.
- Moreover, in the finite element method, it will often be possible to improve or refine the approximate solution by spending more computational effort.

Finite Element Method (cont…)

- Finite Element, Assemblage, Nodes or Nodal Point
- Used to solve physical problems involving complicated geometries, loading and material properties.
- Based on application finite elements are classified as follows
    - Structural Problems
    - Non Structural Problems





F
F
$U_x=0$
$U_y=0$

Common FEA Applications

- **Mechanical/Aerospace/Civil/Automotive Engineering**
- **Structural/Stress Analysis**
    - **Static/Dynamic**
    - **Linear/Nonlinear**
- **Fluid Flow**
- **Heat Transfer**
- **Electromagnetic Fields**
- **Soil Mechanics**
- **Acoustics**
- **Biomechanics**

General Steps in FEM

- Dealing with only structural problems
- Two general methods
  - Force methods
    - Internal forces are considered as the unknowns of the problems.
  - Displacement or Stiffness method.
    - Displacements of nodes are considered as the unknowns of the problems.
- Among two approaches, displacement method is more desirable because its formulation is simpler for most structural analysis problems.

General Steps in FEM

Step: 1 Discretization of structure.

Step: 2 Numbering of nodes and elements.

Step: 3 Selection of proper displacement or interpolation function

Step: 4 Define the material behavior by strain-displacement and stress-strain relationships

Step: 5 Derivation of element stiffness matrix and equations.

Step: 6 Assemble the element equations to obtain the global equations

Step: 7 Applying boundary conditions.

Step: 8 Solution for the unknown displacements

Step: 9 Computation of the element strains and stresses from the nodal displacements

Step: 10 Interpret the results

Three Phases Of Finite Element Method

- Preprocessing
- Analysis
- Post Processing

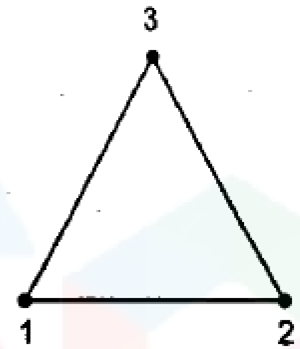
Step: 1 Discretization of structure.

- The art of subdividing a structure in to convenient number of smaller elements is known as Discretization.
- Smaller elements are classified as follows.
  - 1 Dimensional elements
    - Beam and bar elements are considered as 1D
    - The simplest line element also known as linear element has two nodes
  - 2 dimensional elements
    - Triangular and rectangular elements
    - Elements are loaded by forces in their own plane.
    - Simplest 2d elements have corner nodes
  - 3. dimensional elements
    - Common elements are tetrahedral and hexahedral (brick) elements.
    - Simplest 3d elements have corner nodes.

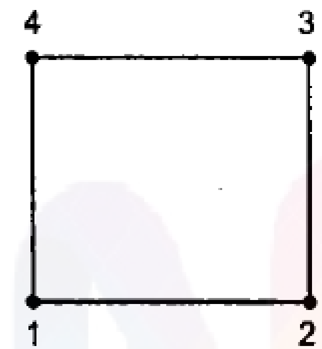
Fig. 1.2. Bar element

1 D element

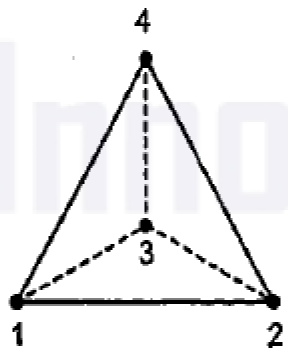
2 D element



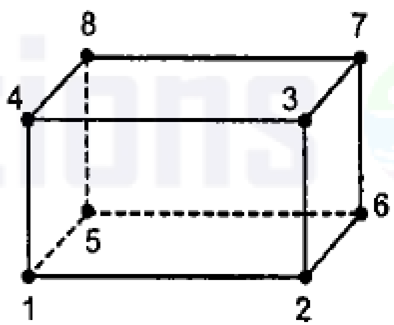
Triangular element



Rectangular element



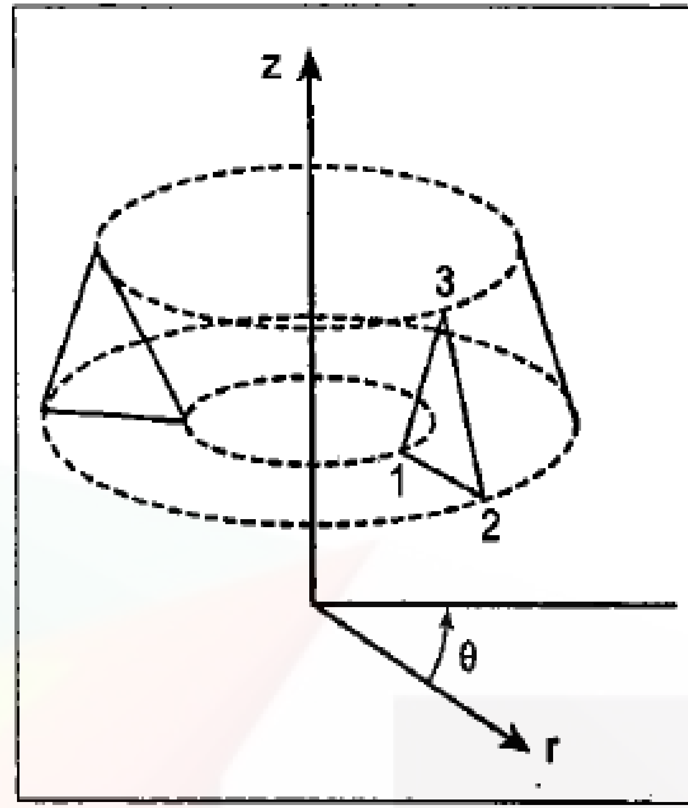
Tetrahedral element



Hexahedral element

3 D element

- Axisymmetric elements
  - Its developed by rotating a triangle or quadrilateral about a fixed axis located in the plane of the element through 360degree.



Step: 2 Numbering of nodes and elements.

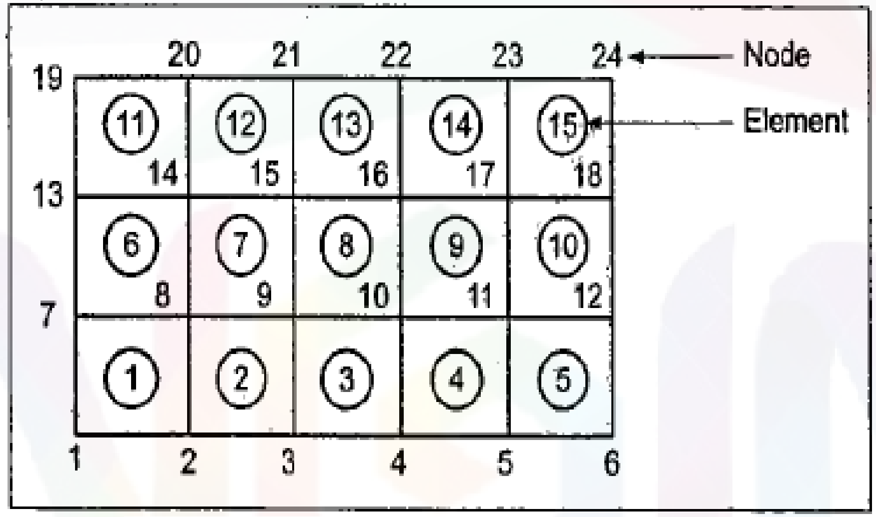
- Nodes and Elements are the very backbones of Finite Element Analysis.
- In FEA, you divide your model into small pieces. Those are called Finite Elements (FE). Those Elements connect all characteristic points (called Nodes) that lie on their circumference. This “connection” is a set of equations called shape functions.
- Each FE has its own set of shape functions, that connect all of the Nodes of that Element). Adjacent Elements share common Nodes (the ones on the shared edge).

- Nodes and elements should be numbered after Discretization process.
- The numbering process is most important since it decide the size of the stiffness matrix and it leads the reduction memory requirement.
- While numbering following condition should be satisfied,

(Maximum node number)	-	(minimum node number)	=	minimum

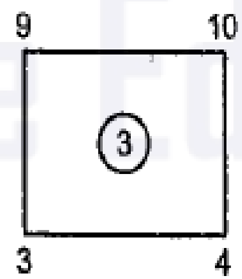
Shorter side numbering is preferred in FEM

Longer Side Numbering Process:



[*Note:* Number with circle denotes element.
Number without circle denotes node]

Considering element (3),

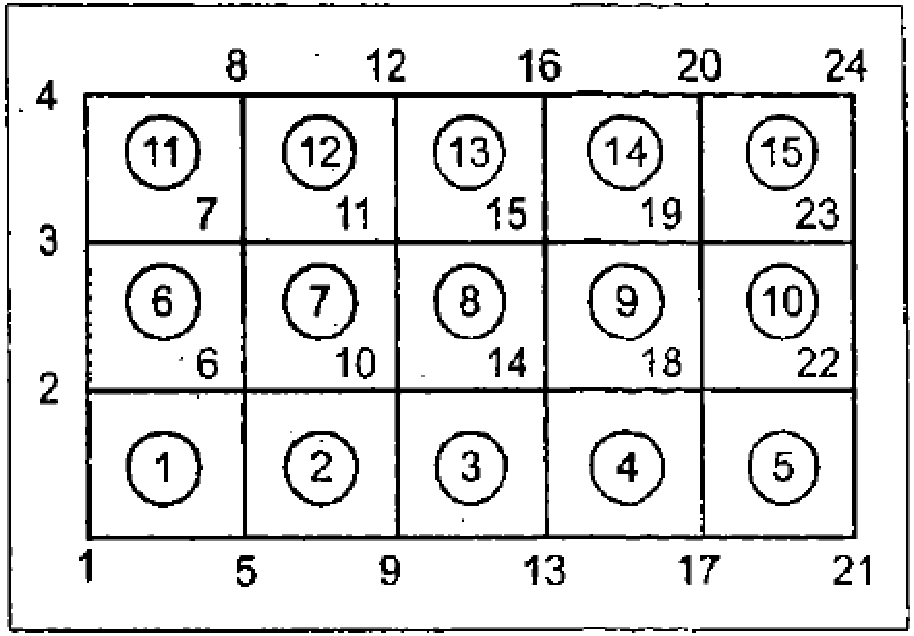


Maximum node number = 10

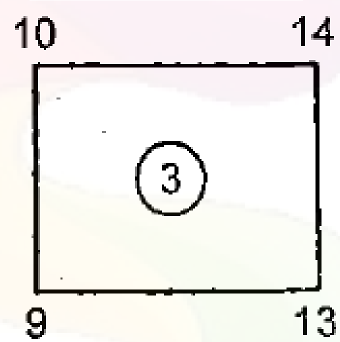
Minimum node number = 3

Difference = 7

Shorter Side Numbering Process:



Considering the same element (3).



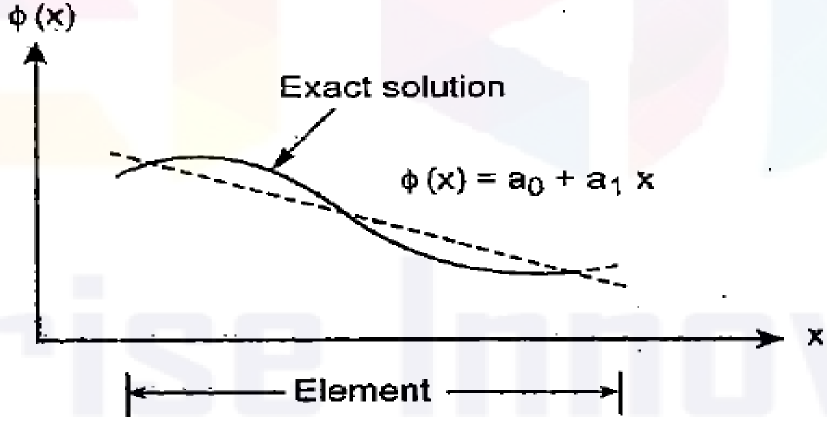
Maximum node number = 14

Minimum node number = 9

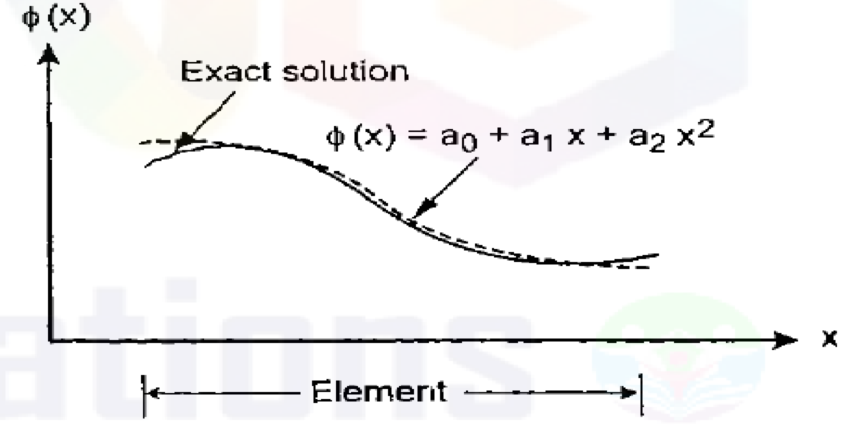
Difference = 5

Step: 3 Selection of proper displacement or interpolation function

- Involves choosing a displacement function within each element.
- Polynomial of linear, quadratic and cubic form are frequently used in displacement function because hey are simper to work
- Reasons for using interpolation functions,
  - Easy to formulate and computerized FE equations
  - Easy to perform differentiation or integration
  - Accuracy of results can be improved by increasing the order of the polynomial.



(a) Linear approximation



(b) Quadratic approximation

Case (i): Linear Polynomial:

$$\text{One dimensional problem } \phi(x) = a_0 + a_1 x$$

$$\text{Two dimensional problem } \phi(x, y) = a_0 + a_1 x + a_2 y$$

$$\text{Three dimensional problem } \phi(x, y, z) = a_0 + a_1 x + a_2 y + a_3 z$$

Case (ii): Quadratic Polynomial:

$$\text{One dimensional problem } \phi(x) = a_0 + a_1 x + a_2 x^2$$

$$\text{Two dimensional problem } \phi(x, y) = a_0 + a_1 x + a_2 y + a_3 x^2 + a_4 y^2 + a_5 xy$$

$$\text{Three dimensional problem } \phi(x, y, z) = a_0 + a_1 x + a_2 y + a_3 z + a_4 x^2 + a_5 y^2 + a_6 z^2 + a_7 xy + a_8 yz + a_9 xz$$

Step: 4 Define the material behavior by strain-displacement and stress-strain relationships

We know that.

By hookes law.

$\sigma \propto e$

$\Rightarrow \sigma = E e \longrightarrow (1)$

also we know,

$\sigma = P/A$

$e = \frac{dl}{l}$ which can be written as

$e = \frac{du}{dx}$

Where $du \rightarrow$ change in ~~dia~~ displacement of element

$dx \rightarrow$ original length of element

$u \rightarrow$ Displacement field variable along x direction.

Step: 5 Derivation of element stiffness matrix and equations.

From Basics. We know that

$$F = K \cdot u$$

$F \rightarrow$ Force
$K \rightarrow$ Stiffness

For 'n' nodes

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ \vdots \\ F_n \end{Bmatrix} = \begin{bmatrix} K_{11} & K_{12} & \cdots & K_{1n} \\ K_{21} & K_{22} & \cdots & K_{2n} \\ K_{31} & \cdots & & K_{3n} \\ \vdots & & & \vdots \\ K_{n1} & K_{n2} & & K_{nn} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ \vdots \\ u_n \end{Bmatrix}$$

in compact matrix form

$$\{F^e\} = [K^e]\{u^e\}$$

$e \rightarrow$ element, $\{F\} \rightarrow$ nodal force vector.
$\{u\} \rightarrow$ element displacement vector
$[K] \rightarrow$ element stiffness matrix

Any method can be used to derive the above equation

Step: 6 Assemble the element equations to obtain the global equations

- Individual element equations obtained in previous steps are added by using a method of superposition
- The final assembled or global equation which is of the form of

$$\{F\}=[k]\{u\}$$

{F}= Global Force vector

[k]= Global stiffness matrix

{u}= Global displacement vector

Step: 7 Applying boundary conditions.

- **[k]** is a singular matrix because its determinant is equal to zero.
- In order to remove the singularity problem, certain boundary conditions are applied so that the structure remains in place instead of moving as a rigid body.
- The global equation to be modified to account for the boundary conditions of the problem.

Step: 8 Solution for the unknown displacements

- A set of simultaneous algebraic equations formed in step 6 can be written in expanded matrix.

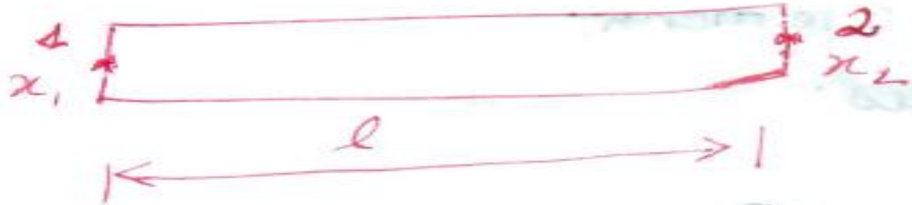
$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ \vdots \\ F_n \end{Bmatrix} = \begin{bmatrix} K_{11} & K_{12} & \cdots & K_{1n} \\ K_{21} & K_{22} & \cdots & K_{2n} \\ K_{31} & \cdots & \cdots & K_{3n} \\ \vdots & & & \vdots \\ K_{n1} & K_{n2} & & K_{nn} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ \vdots \\ u_n \end{Bmatrix}$$

- These equation is solved and unknown displacements are calculated using Gauss-Seidel method or Gaussian elimination method.

Step: 9 Computation of the element strains and stresses from the nodal displacements

$$\text{Strain} = \frac{du}{dx}$$

$$e = \frac{u_2 - u_1}{x_2 - x_1}$$



u_1 & u_2 are displacement at node 1 & 2

$x_2 - x_1$ = actual length (l) of the element

From this we can find the strain value.

& the stress can be found by

$$\sigma = E e$$

Step: 10 Interpret the results

- Analysis and evaluation of the solutions results is referred to the post processing.
- Post processing computer programs helps the user to interpret the results by displaying them in graphical form.