



A T M E
College of Engineering



Finite Element Method BME701

Module-1 **Introduction to Finite Element Method**

Mr. SUKRUTH SAGAR B P
Assistant Professor,
Dept. of Mechanical Engineering,
ATMECE, Mysuru.

MODULE-1

Introduction to FEM:

Introduction to FEM,
engineering applications,
advantages,
General steps,
Element types,
Convergence criteria,
Coordinate systems,
commercial packages,
Preprocessor, solver and post processor.
Principles of Elasticity: Strain displacement relations,
Stress strain relations for 1D, 2D, and 3D cases,
Plain stress and Plain strain conditions,



Introduction to Numerical Methods, Rayleigh-Ritz method,
Potential energy method and
Galerkin method applied to simple problems on axially loaded members, cantilever, simply supported beams, with point loads and distributed loads.

Introduction to FEM:

There are 3 methods to solve an engineering problem

- Analytical Method
- Experimental Method
- Numerical Method

1. Analytical method:

- It consists of equations basically derived from assumptions and it is used to solve simple geometry, loading and boundary conditions. It gives accurate results.
- The limitation of analytical methods is complex problems
- Ex: Strength of Materials, Theory of elasticity

2. **Experimental method:**

- It consists of instruments, equipment to carry analysis. It gives accurate results only if the equipment are properly calibrated.
- The limitations for experimental methods is complex problems.
- Ex: Photo elasticity, Strain gauges

3. **Numerical methods:**

- These are the most powerful methods, to analyse the complex problems, involving complicated boundary conditions and loading.
- These will give appropriate results, not involve any assumptions.
- Ex: Finite Element Method, Boundary Element Method, Finite Difference Method

Finite Element Method:

- It is a numerical tool used to solve complex engineering problem.
- The physical domain of a problem is divided into number of sub domain, each sub domain is called Finite Element.
- The finite element in the structure establishes a connection with the adjacent finite element through a certain discrete points called **Nodes**.
- The quantities of interest such as displacement, temperature and pressure called **Field Variables**.

Advantages of FEM:

1. This method can be applied to any engineering problems, which are complex in nature.
2. This method can be systematically programmed to accommodate complex and difficult problems having non-homogeneous materials, non-linear stress-strain behavior and complicated boundary conditions.
3. This method possesses variety of levels at which we may develop an understanding of the technique i.e. the learner can understand this technique through simple one-dimensional structure problem and can extend it to two- and three-dimensional problems.

4. The finite element mesh can mix elements of different type, shapes and physical properties.
5. User prepared input data controls the selection of problem type, geometry, boundary condition, element selection etc.
6. There is a close physical resemblance between the actual structure and its finite element model.

Disadvantages of FEM:

1. A computer, a reliable program and intelligent user are essential to get correct result.
2. A general-purpose program has extensive documentations which cannot be ignored.
3. An experience and good engineering judgments are needed in order to define a good model.
4. Many input data are required and voluminous output data must be stored and understood.

Applications of FEM

The applications of FEM, in general, ranges from deformation and stress analysis of automotive, aircraft, building and bridge structures to field analysis such as heat flux, fluid flow, magnetic flux, seepage and other flow problems.

Some examples of specific applications of FEM for the important boundary value problems are as listed below:

1. Structural engineering, structural mechanics and aerospace engineering.
Ex.: (a) Analysis of beam, plate and shell structures.
(b) Analysis of complex structures.
(c) Two and three - dimensional stress analysis.
(d) Torsion of prismatic bar.

2. Soil mechanics, foundation engineering and rock mechanics.

Ex.: (a) Two and three - dimensional stress analyses.

(b) Construction and excavation problems.

(c) Soil structure interaction.

(d) Analysis of dams, tunnels, bore wells, etc.

(e) Steady state seepage of fluids in soils and rocks.

3. Heat conduction.

Ex.: Steady state temperature distribution in solids and fluids.

4. Hydrodynamic, hydraulic engineering and water resources.

Ex.: (a) Solution for potential flow of fluids.

(b) Solutions for viscous flow of fluids.

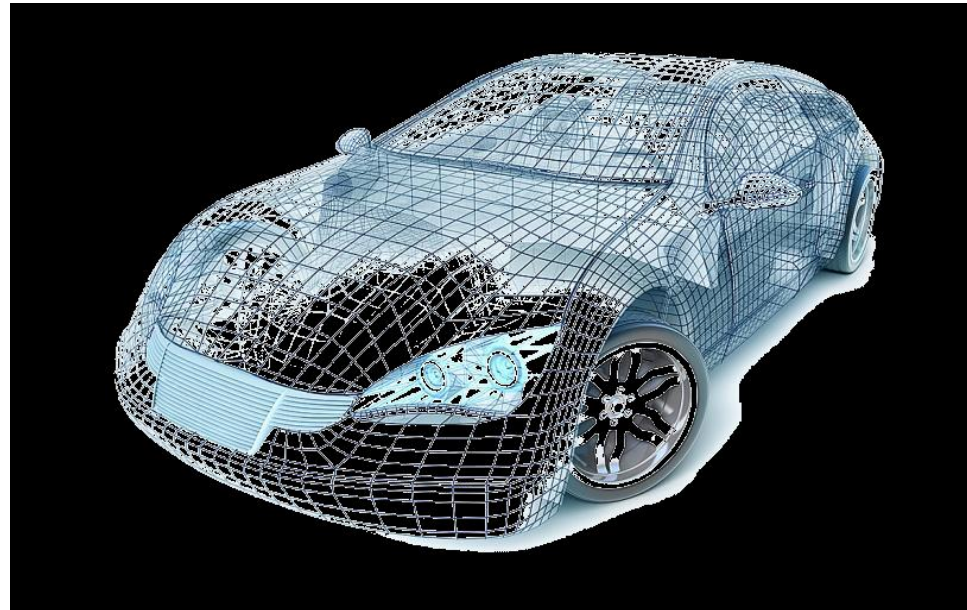
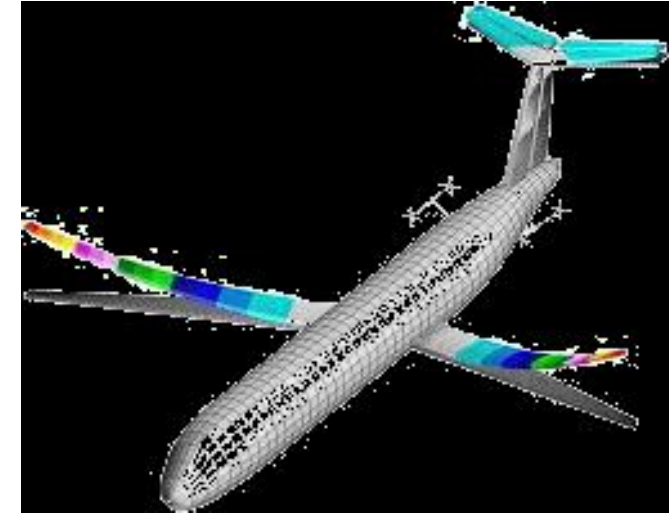
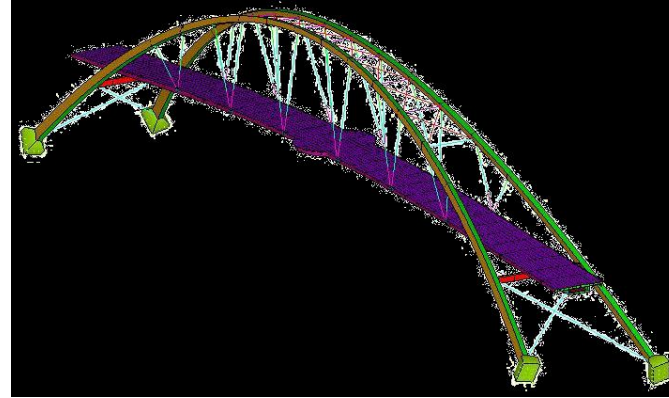
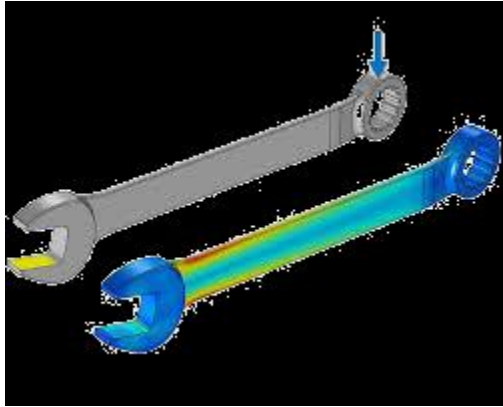
(c) Analysis of hydraulic structures and dams.

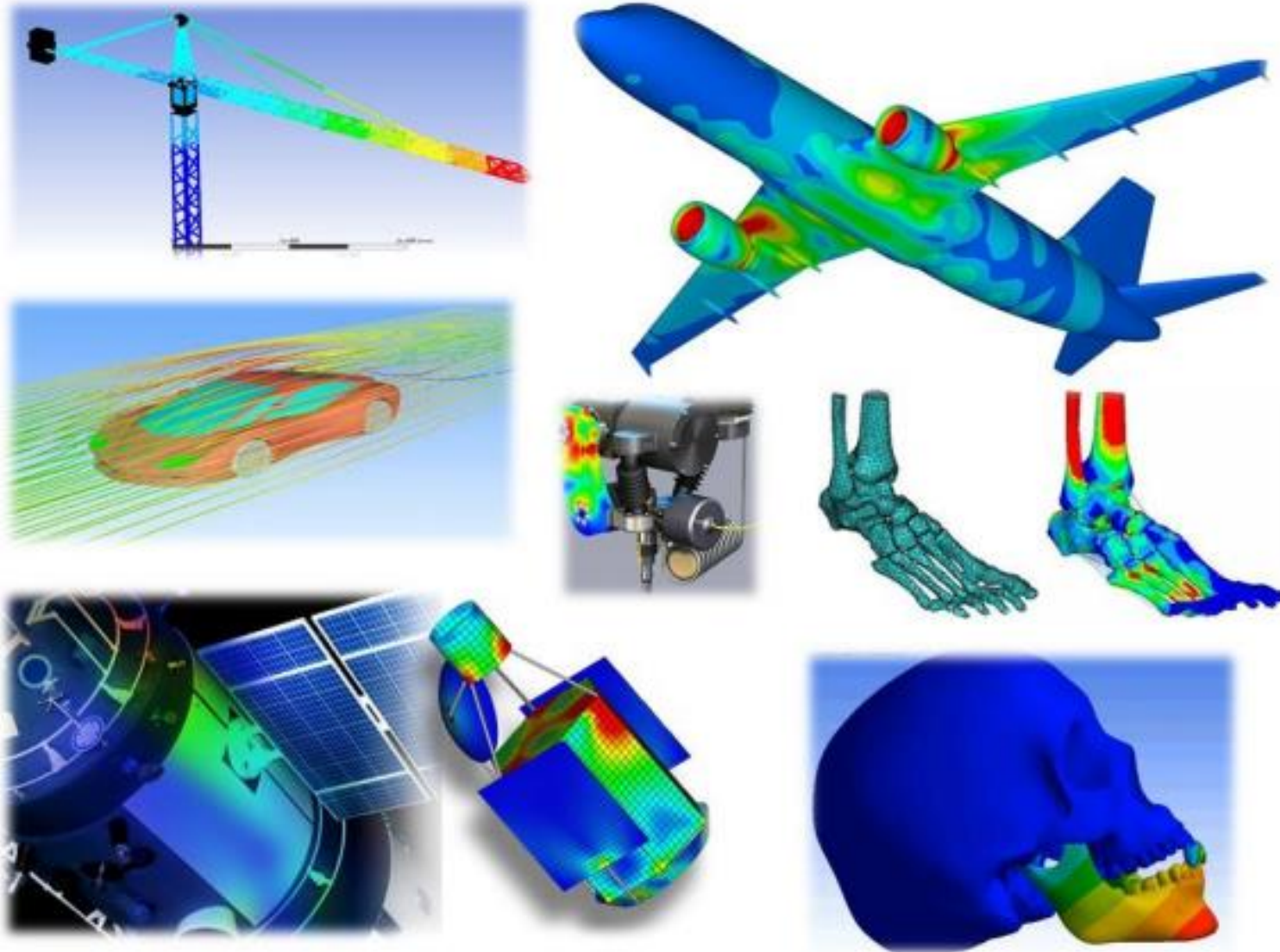
5. Nuclear engineering,

Ex.: (a) Analysis of reactor containment structures.

(b) Steady state temperature distribution in reactors and reactor structures.

Engineering applications of Finite Element Method





General Steps in FEM:

The following are basic steps used in FEM,

1. **Discretization of the continuum.**
2. **Selection of displacement model.**
3. **Derivation of element stiffness matrix.**
4. **Assembly of algebraic equations.**
5. **Boundary conditions.**
6. **Solution of the unknown displacement.**
7. **Computation of element strains and stresses.**

1. Discretization of the continuum:

- *Discretization* is the process of subdividing a given continuum into finite elements as shown in Fig. 1.
- The finite elements may be triangular or quadrilateral etc. for a two-dimensional problems; they may be tetrahedron or rectangular prism etc. for three-dimensional problems.

The selection of finite elements in FEM is depends on the following factors,

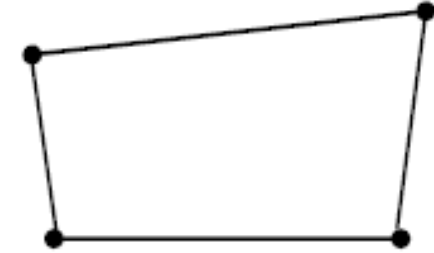
1. Dimension of the problem.
2. Shape of the body or continuum.
3. Accuracy of the results required.
4. Cost and time for the analysis.



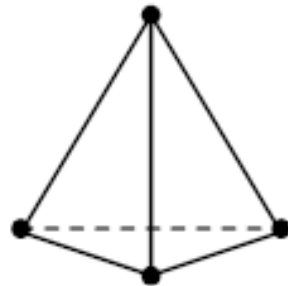
(a) Line element
for 1 – D problem



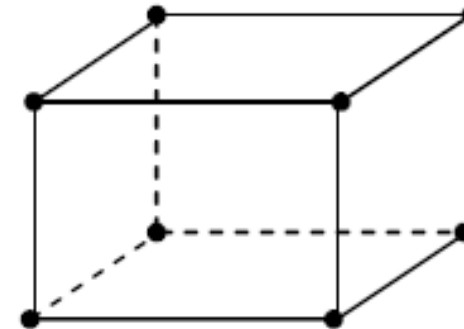
(b) 3-noded triangle
for 2 – D problem



(c) 4-noded quadrilateral
for 2 – D problem



(d) 4-noded tetrahedron
for 3 – D problem



(e) 8-noded rectangular prism
for 3 – D problem

Fig.: Typical finite elements

To illustrate the concept of step-by-step procedures, consider a stepped column shown in Fig. 8 (a) as an example problem. Fig. 8 (b) shows the discretization of the stepped column into two elements. Let $l_1 = l_2 = 1m$; $A_1 = 20m^2$, $A_2 = 10m^2$ and $E_1 = E_2 = 2 * 10^{11} N / m^2$.

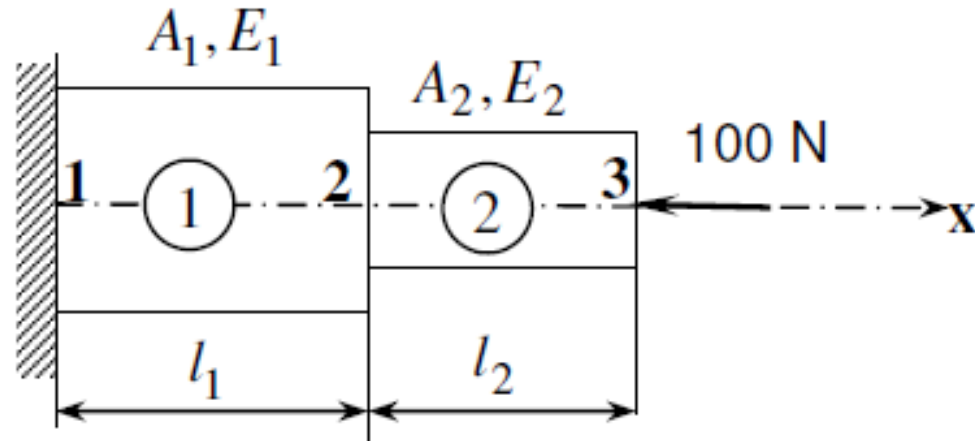


Fig. 8 (a): Stepped column

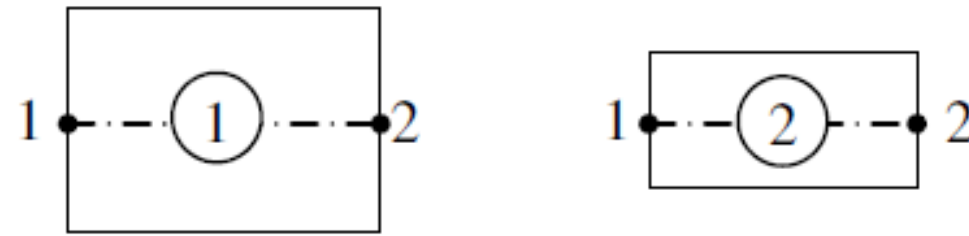


Fig. 8(b): Discretization

2. Selection of displacement model:

- Simple functions are assumed to approximate the displacements over each element.
- These functions are called displacement models or displacement functions.
- The trigonometric or polynomial functions are assumed for these displacement models.
- The polynomial functions are much easier for mathematical operation and hence they are widely used in FEM.

The selection of displacement model is based on the following three factors;

1. The type and degree of the displacement model.
2. The particular displacement magnitude that describe the model.
3. The model should satisfy certain requirements.

3. Derivation of element stiffness matrix:

- The stiffness matrix consists of the coefficients of the equilibrium equations derived from the material and geometric properties of an element.
- This matrix relates the displacements at the nodal points (nodal displacement) to the nodal forces.

- In general, using stiffness matrix the equilibrium equation for an e th element shown in Fig. 9 can be written as

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

where,

$$[K]^e = \frac{A_e E_e}{l_e} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}, \text{ element stiffness matrix}$$

$$\{q\}^e = \{u_i^e \quad u_j^e\}^T, \text{ nodal displacement vector and}$$

$$\{F\}^e = \{P_i^e \quad P_j^e\}^T, \text{ nodal load vector.}$$

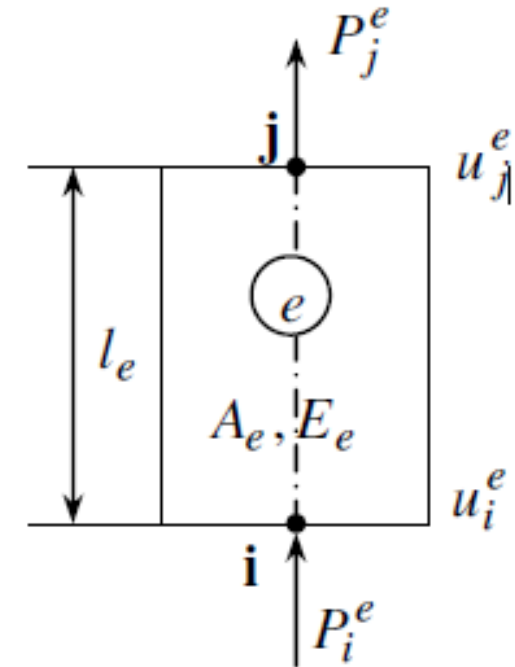


Fig. 9: Typical element

4. *Assembly of algebraic equations:*

- The assembly process includes the assembly of the overall or global stiffness matrix for the entire body from the individual element stiffness matrices and the global load vector from the element nodal force vector.
- In general, the overall equilibrium equation between the global stiffness matrix, global displacement vector and global load vector is expressed as $[K]\{Q\} = \{F\}$

$$\begin{bmatrix} \frac{A_1 E_1}{L_1} & -\frac{A_1 E_1}{L_1} & 0 \\ -\frac{A_1 E_1}{L_1} & \left(\frac{A_1 E_1}{L_1} + \frac{A_2 E_2}{L_2} \right) & -\frac{A_2 E_2}{L_2} \\ 0 & -\frac{A_2 E_2}{L_2} & \frac{A_2 E_2}{L_2} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} R_1 \\ 0 \\ -F \end{Bmatrix}$$

- For the example problem considered, the global stiffness matrix $[K]$ can be assembled using the following connectivity table.

Element No.	Local node Nos.	Global node Nos.
1	1	1
	2	2
2	1	2
	2	3

Write the global node numbers for the corresponding element stiffness matrices and assemble the matrices we get global stiffness matrix as

$$[K] = 10^{12} \begin{bmatrix} 1 & 2 & 3 \\ 4 & -4 & 0 \\ -4 & 4+2 & -2 \\ 0 & -2 & 2 \end{bmatrix} \begin{matrix} 1 \\ 2 \\ 3 \end{matrix}$$

The global nodal force vector becomes as

$$\{F\} = \begin{Bmatrix} P_1 \\ P_2 \\ P_3 \end{Bmatrix} = \begin{Bmatrix} R \\ 0 \\ -100 \end{Bmatrix}$$

Reaction at fixed node 1
As there is no external load at node 2
As external load at node 3 acts in - ve x - direction

Global nodal displacement vector becomes as

$$\{Q\} = \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}$$

Using the above three matrices, the global equilibrium equation can be written as

$$10^{12} \begin{bmatrix} 4 & -4 & 0 \\ -4 & 6 & -2 \\ 0 & -2 & 2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} R \\ 0 \\ -100 \end{Bmatrix} \quad \text{--- (A)}$$

5. Boundary conditions:

- After assembling the elemental equilibrium equations, the boundary conditions are to be applied and the overall equilibrium equation must be modified depending upon the available boundary conditions.

The overall equilibrium equation can be modified for given boundary conditions using one of the following methods.

1. Elimination method.
2. Penalty method and
3. Multi constraint method.

- In our example problem, as node 1 is fixed, $u_1=0$.
- Therefore, using elimination method the overall system equation i.e. Eqn. (A) can be modified by eliminating 1st row and 1st column. It is given by

$$10^{12} \begin{bmatrix} 6 & -2 \\ -2 & 2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -100 \end{Bmatrix} \quad \text{--- (B)}$$

6. *Solution of the unknown displacement:*

- The simultaneous equations obtained in step 5 are solved for the unknown displacements.
- Large sets of simultaneous equations are solved using Gauss elimination method.
- For the example problem, the solution of Eqn. (B) yields nodal displacements u_2 and u_3 .

They are obtained as follows:

$$\begin{aligned} 6 * 10^{12} u_2 - 2 * 10^{12} u_3 &= 0 \\ -2 * 10^{12} u_2 + 2 * 10^{12} u_3 &= -100 \end{aligned}$$

Adding two Eqs. $4 * 10^{12} u_2 = -100$

Therefore, $u_2 = -25 * 10^{-12}$ m. Substituting u_2 into any one of the above Eqns. we get

$$u_3 = -7.5 * 10^{-12} \text{ m.}$$

7. Computation of element strains and stresses:

- Using the nodal displacements, the elemental strains and stresses can be determined as follows:

In general, the strain in e th element can be expressed as

$$\varepsilon_e = \frac{u_j - u_i}{l_e} \text{ Then, the stress is expressed as } \sigma_e = E_e \varepsilon_e$$

Therefore, the strains and stresses for the problem considered are:

$$\text{For element 1: Strain, } \varepsilon_1 = \frac{u_2 - u_1}{l_1} = \frac{-25 * 10^{-12} - 0}{1} = -25 * 10^{-12} \text{ (compressive)}$$

$$\text{Stress, } \sigma_1 = E_1 * \varepsilon_1 = 2 * 10^{11} * -25 * 10^{-12} = -5 \text{ N/m}^2$$

$$\text{For element 2: Strain, } \varepsilon_2 = \frac{u_3 - u_2}{l_2} = \frac{-75 * 10^{-12} - (-25 * 10^{-12})}{1} = -50 * 10^{-12} \text{ (compressive)}$$

$$\text{Stress, } \sigma_2 = E_2 * \varepsilon_2 = 2 * 10^{11} * -50 * 10^{-12} = -10 \text{ N/m}^2$$

Element types

Type of Elements or Various Elements Shapes or Element Library:

- The discretization of the structure or body into finite elements forms the first step in finite element analysis of a complicated structural system.
- These elements coincide with the geometry of the structure and represent the geometry and mechanical properties in the regions.
- Therefore, the first decision the engineer must make is to select the shape of the basic elements to be used in the analysis.

The selection of finite element in FEM is depends on the following factors.

1. Dimension of the problem.
2. Shape of the body or continuum.
3. Accuracy of the results required.
4. Cost and time for the analysis.

Fig. shows some typical elements used to discretize the continuum.

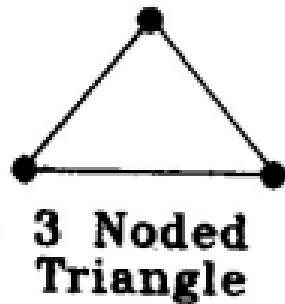


(a) 2-noded element

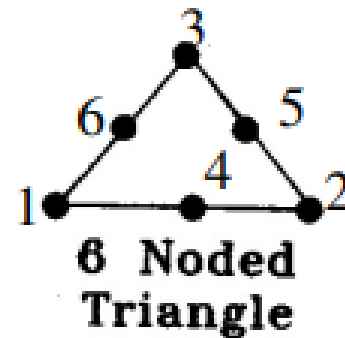


(b) 3-noded element

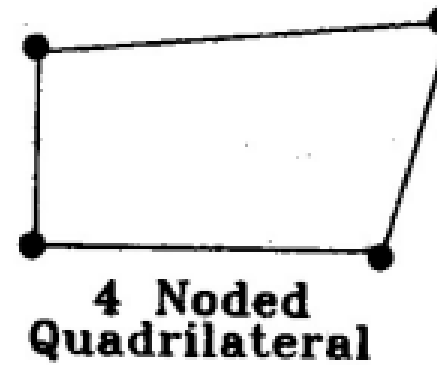
(a) One – dimensional bar element



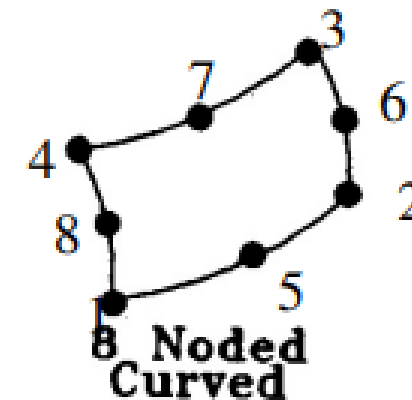
(a)



(b)

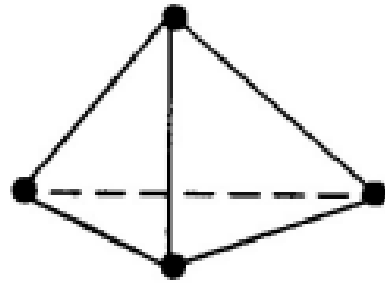


(c)

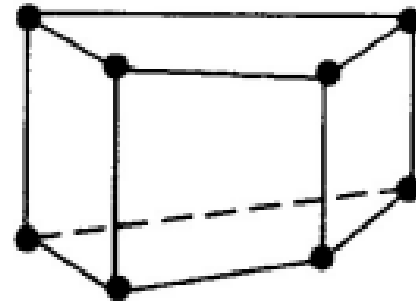


(d)

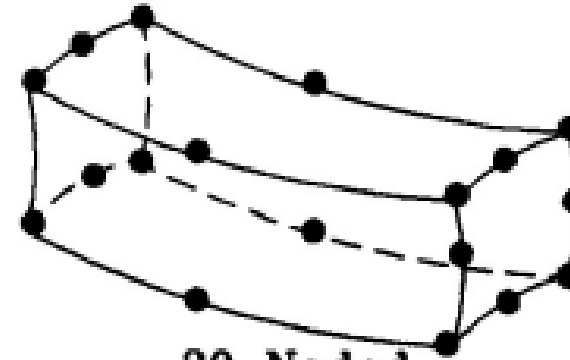
(b) Two – dimensional elements



**4 Noded
Tetrahedron**



**8 Noded
Hexahedron**



**20 Noded
Curved Solid**

(c) Three – dimensional elements

1D Elements: When the geometry, material properties and field variable of the problem is described in terms of only one spatial coordinate (for example x-coordinate), then line elements shown in Fig. (a) can be used.

2D Elements: When the configuration and other parameters of the problems are described in terms of two spatial coordinates, then two-dimensional elements are used. Triangular or quadrilateral elements (Fig.1b) are used for two – dimensional problems such as plane stress or plain strain or plate bending problems.

3D Elements: When the geometry, material properties and other parameters of the problems are described by three spatial coordinates, then three-dimensional elements shown in Fig. 1(c) are used.

Convergence Criteria:

In FEM sequence of approximate solutions are obtained as the element size is reduced successively. These solutions converge to the exact solutions if the interpolation polynomial satisfies the following convergence requirements:

1. *The displacement model must be continuous within the element and the displacement must be compatible between adjacent elements:* The condition can be satisfied by choosing polynomials which are inherently continuous functions for the displacement model. The second condition indicates that when the elements deform there should not be any discontinuity between the elements (elements should not overlap or separate).

2. *The displacement model must be capable of representing rigid body displacement of the element:* This condition states that when the nodes are given such displacements corresponding to a rigid body motion, the element should not experience any strains and hence leads zero nodal forces. The constant term present in the polynomial functions satisfy this condition.

3. *The displacement model must be capable of representing constant strain state within the element:* When the elements are made smaller and smaller size, the strain in each elements approach constant value. In such case, the assumed displacement model should include the terms to represent the constant strain state within the element. For one, two and three - dimensional elasticity problems, the linear terms (i.e. $2a$) present in the polynomial functions satisfy this requirement.

In order to satisfy the above mentioned three requirements, the displacement model must contain at least constant and linear terms in the polynomials.

Coordinate systems

The following coordinate systems are used in FEM.

1. Global coordinate system
2. Local coordinate system
3. Natural coordinate system

Global coordinate system: The coordinate system used to define entire body is called global coordinate system. Fig. 8(a) shows the cartesian global coordinate system used for some of the typical elements.

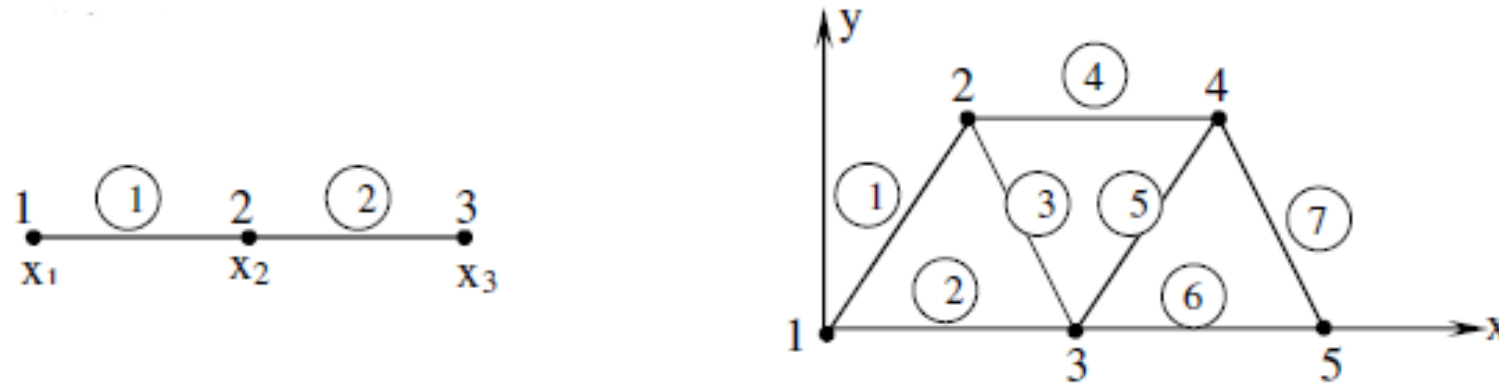


Fig. 8(a): Global coordinate system

Local coordinate system: A *local coordinate system* is one that is defined for a particular element and not for the entire body.

- This local coordinate system simplifies the derivation of element formulations. For example, for typical truss elements shown in Fig. 8(a), the local coordinates (x, y) may be as shown in Fig. 8(b).
- However, for this truss problem the final equations derived in terms of local coordinate are to be transformed to the common global coordinate system.

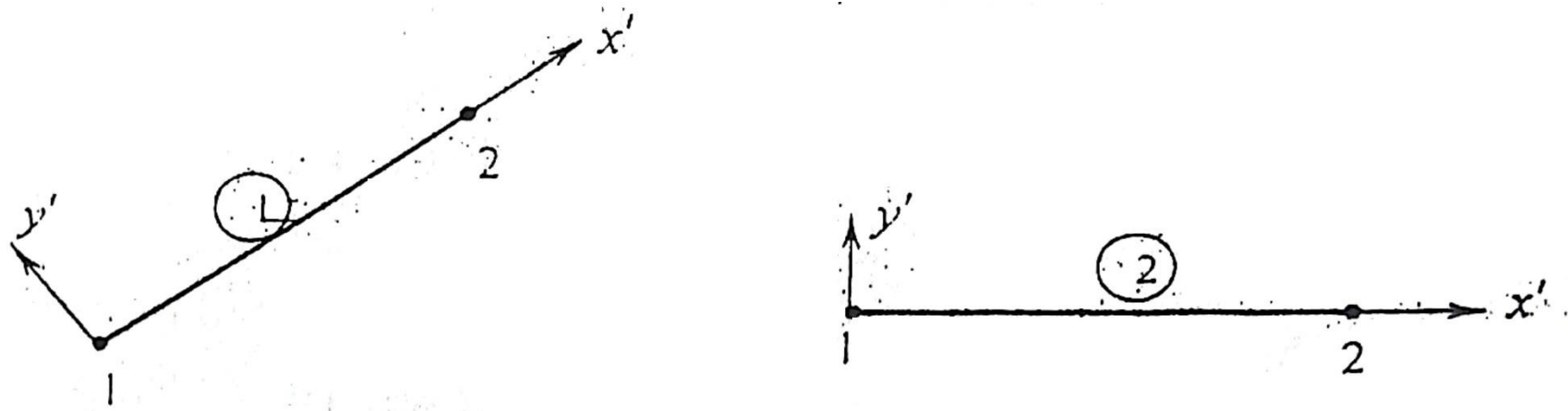
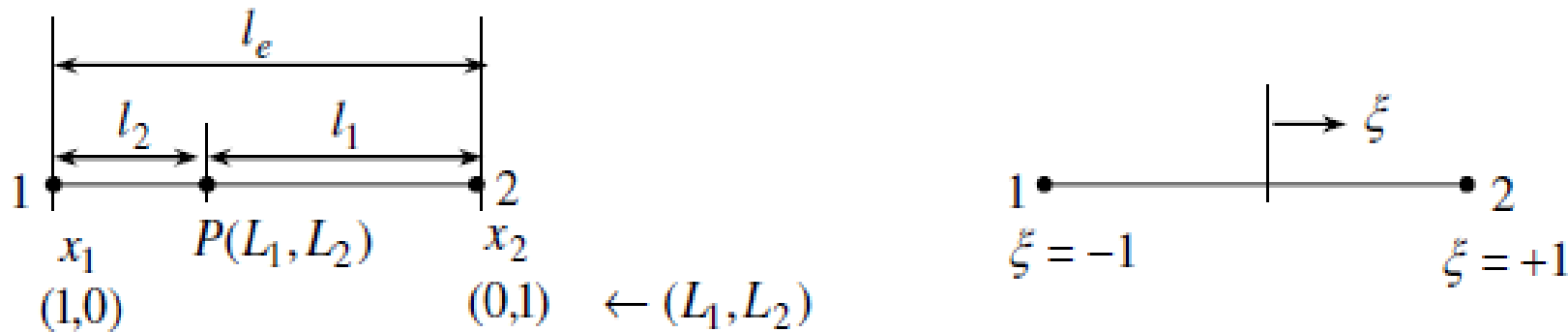


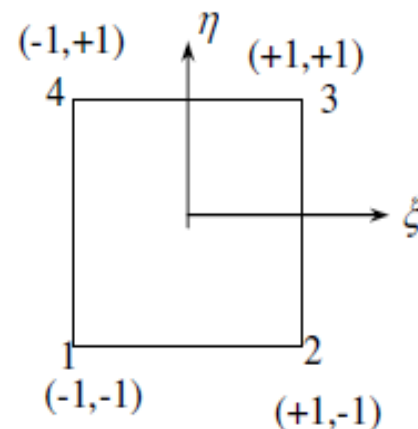
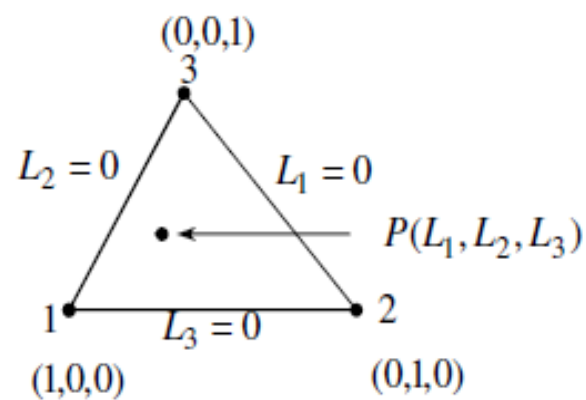
Fig. 8(b): Local coordinate system

Natural Coordinate Systems: The *natural coordinate system* is a local system used to specify the position of a point within the element by a set of dimensionless numbers.

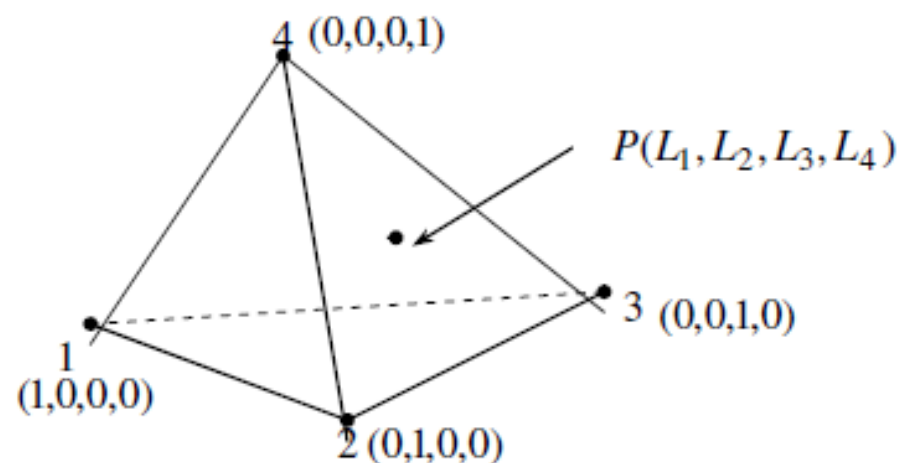
- The magnitudes of these number never exceed unity. Natural coordinate systems are so defined that, their values are zero or one at the nodal points of an element.
- This coordinate system facilitates the integration while computing element stiffness matrix. Fig. 8 shows some of the natural coordinate systems used for one, two and three – dimensional elements.



(a) For 1 – D bar elements



(b) For 2 – D elements



(c) For 3 – D element

Fig. 8: Natural coordinate systems

Commercial packages in FEM

➤ General-Purpose & Multi-physics:

- **ANSYS:** A comprehensive suite widely used across aerospace, automotive, and electronics. Its module ANSYS Mechanical handles everything from linear static stress to fluid-structure interaction (FSI) and electromagnetics.
- **SIMULIA Abaqus:** Known for its mature nonlinear capabilities and complex contact modeling (e.g., severe deformation, drop tests).
- **COMSOL Multi-physics:** The go-to tool for deeply coupled, multiphysics problems—such as Joule heating, acoustics, and chemical reactions.

➤ Aerospace & Mechanical:

- **Siemens Simcenter (Femap & Nastran):** Femap serves as the pre- and post-processor, while Nastran is the powerful industry-standard solver. It is heavily utilized in aerospace, defense, and machinery.
- **LS-DYNA:** Renowned for explicit dynamic analysis, making it the premier tool for automotive crash testing, bird strikes, and high-velocity impacts

Preprocessor, solver and post processor

- A Finite Element Method (FEM) workflow is divided into three fundamental stages: the **Pre-processor** (model setup), the **Solver** (computation), and the **Post-processor** (result visualization).
- Together, these steps transform a physical engineering problem into a solvable mathematical model and ultimately into actionable data.

1. Pre-processor

The pre-processor is the user interface where the model is built and prepared for analysis.

2. Solver

The solver acts as the "calculation engine" of the FEM software. It reads the input file created by the pre-processor and performs no graphics work.

3. Post-processor

The post-processor takes the raw numerical data produced by the solver and translates it into a graphical format that engineers can easily interpret.

Principles of Elasticity: Strain - displacement relations

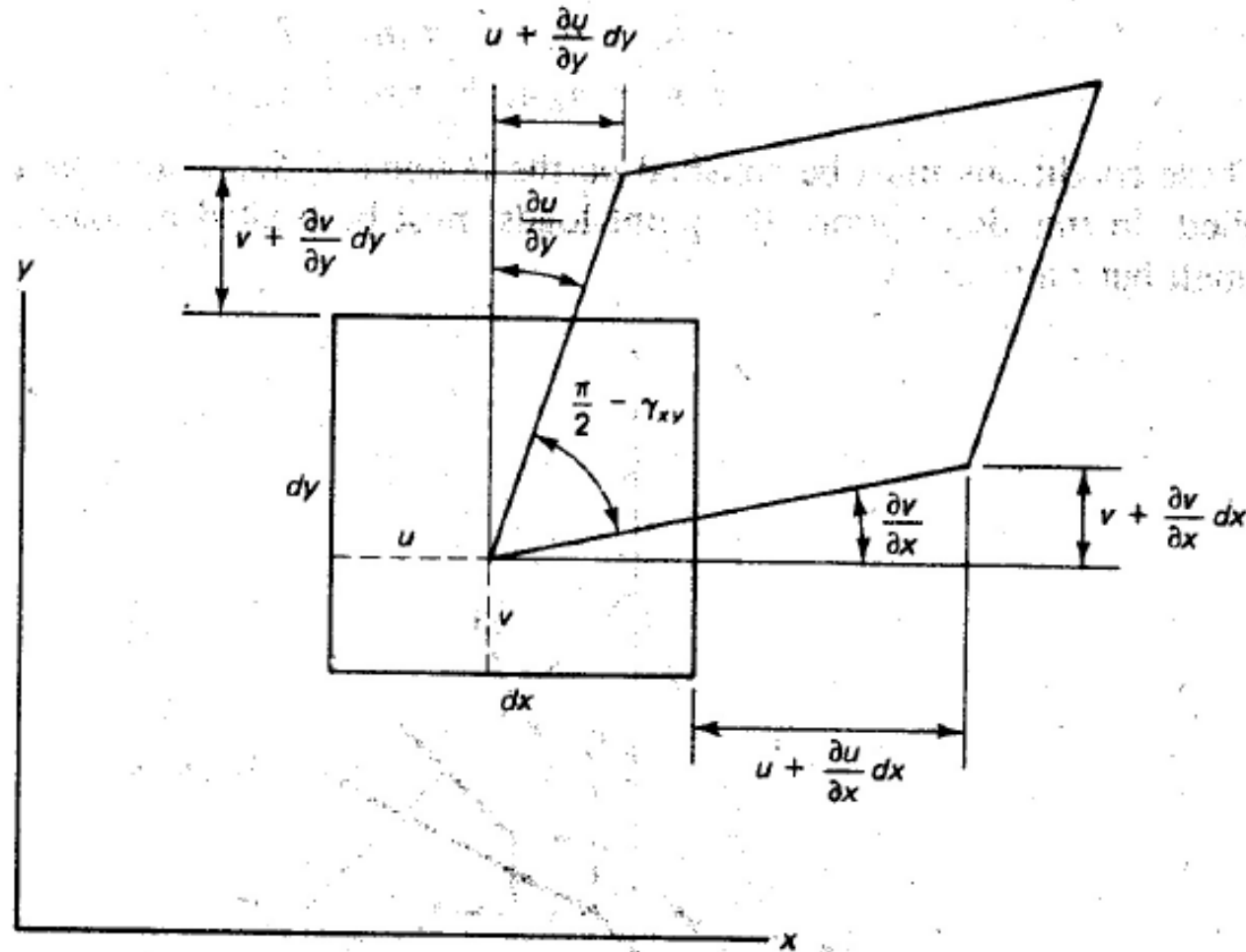


Fig. 5: Deformed elemental surface

- Consider a plane ABCD before loading as shown in Fig. 5. As a result of loading, the plane occupies the position and shape as shown.
- By definition, the normal stress is the ratio of change in length to original length. Therefore, the strain in x – direction is given by,

$$\epsilon_x = \frac{\text{Change in length}}{\text{Original length}} = \frac{u + \frac{\partial u}{\partial x} dx - u}{dx} = \frac{\partial u}{\partial x} \quad \text{---(1)}$$

Similarly,

$$\epsilon_y = \frac{\partial v}{\partial y} \quad \text{---(2)}$$

$$\epsilon_z = \frac{\partial w}{\partial z} \quad \text{---(3)}$$

The shear strain is defined as the amount of change in right angle of plane. Therefore,

$$\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \quad \text{---(4)}$$

$$\gamma_{yz} = \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \quad \text{---(5)}$$

$$\gamma_{zx} = \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \quad \text{---(6)}$$

Stress strain relations for 1D, 2D, and 3D cases

- The stress – strain relation for an isotropic linear elastic material can be obtained from the generalized Hooke's law.
- For an isotropic material, the two material properties namely Young's modulus ' E ' and Poison's ratio ' μ ' are required to describe the stress – strain relation.
- Consider a elemental cube inside a body. The strains in x, y and z directions can be expressed as

$$\varepsilon_x = \frac{\sigma_x}{E} - \mu \frac{\sigma_y}{E} - \mu \frac{\sigma_z}{E} \quad \text{---(1)}$$

$$\varepsilon_y = -\mu \frac{\sigma_x}{E} + \frac{\sigma_y}{E} - \mu \frac{\sigma_z}{E} \quad \text{---(2)}$$

$$\varepsilon_z = -\mu \frac{\sigma_x}{E} - \mu \frac{\sigma_y}{E} + \frac{\sigma_z}{E} \quad \text{---(3)}$$

and shear strains as

$$\gamma_{xy} = \frac{\tau_{xy}}{G} = \frac{2(1+\mu)}{E} \tau_{xy} \quad \text{---(4)}$$

$$\gamma_{yz} = \frac{\tau_{yz}}{G} = \frac{2(1+\mu)}{E} \tau_{yz} \quad \text{---(5)}$$

$$\gamma_{zx} = \frac{\tau_{zx}}{G} = \frac{2(1+\mu)}{E} \tau_{zx} \quad \text{---(6)}$$

$$\therefore G = \frac{E}{2(1+\mu)}$$

The above equations can also be written in matrix form as

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

In short form

$$\{\varepsilon\} = [C]\{\sigma\} \quad \text{---(A)}$$

The above equation relates stress with strain i.e. if six components of stresses are known then six components of strains can be determined from the above equation. It can also be possible to relate the strain with stress by solving Eqns. (1) to (6) and is expressed as,

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

In short form
 $\{\sigma\} = [D]\{\epsilon\}$

---(B)

where,

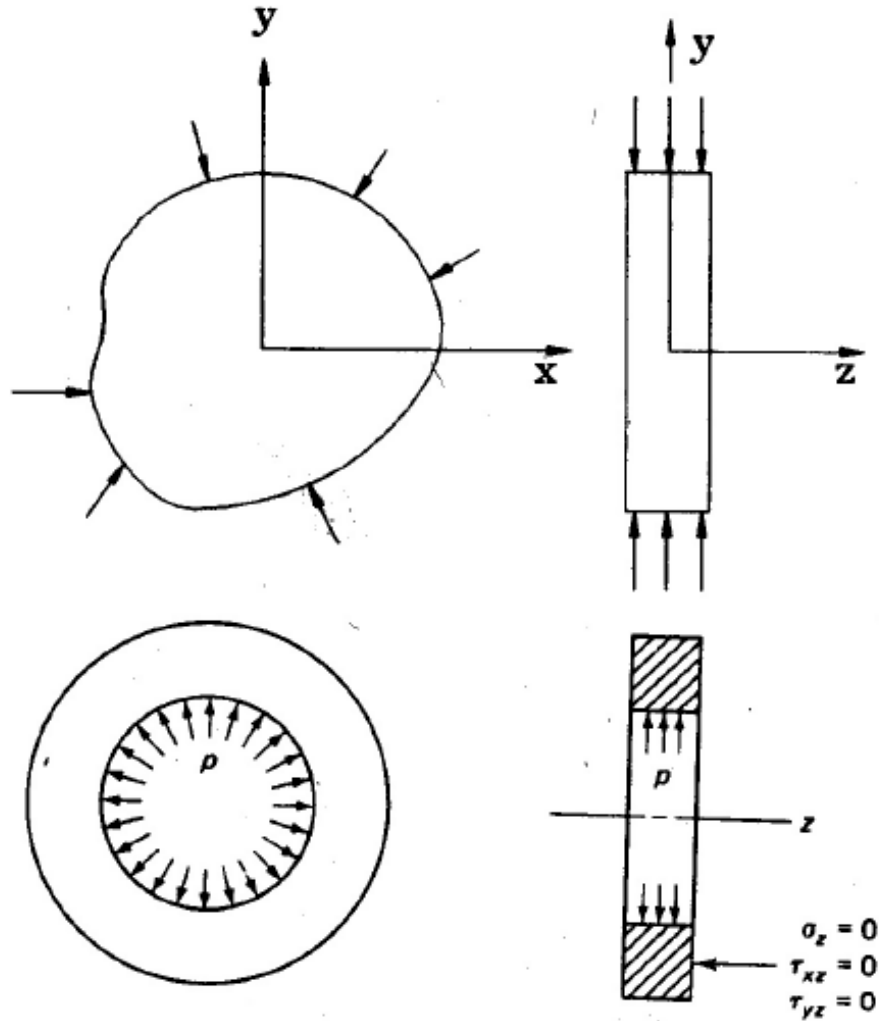
The matrices **C** and **D** in Eqns. (A) and (B) are called as *constitutive or material or elasticity matrix*.

The matrix **C** in Eqn. (A) relates stress to strain and matrix **D** in Eqn. (B) relates strain to stress and these matrices are square, symmetric matrices.

Plain stress and Plain strain conditions

- The 3 – D problems in solid mechanics can be analyzed using FEM.
- In such cases computational time increases due to more number of equations and hence analysis becomes costlier.
- However, there are many situations where geometry and loading configurations permit to reduce these 3 – D problems into 2 – D problems and accurate results can be obtained.
- These 2 – D problems are formulated as
 - (i) plane stress
 - (ii) plane strain and
 - (iii) axi – symmetric problems.

1. Plane Stress Problems:



(a) Thin plate with in – plane loading

(b) Ring press fitted on a shaft

Fig. 6: Plane stress problems

- The problems involving a very small dimension in z – direction are referred as plane stress problems.
- The examples of plane stress problems are a thin plate with in – plane loading (Fig. 6(a)) and a ring press fitted on a shaft (Fig. 6(b)).
- In this problem there is no load applied on the surface of the plate i.e. in z – direction. Hence, the stress components in z – direction is zero.

$$\text{i.e. } \sigma_z = \tau_{xz} = \tau_{yz} = 0$$

The stress – strain relation for this plane stress problem reduces to

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{(1-\mu^2)} \begin{bmatrix} 1 & \mu & 0 \\ \mu & 1 & 0 \\ 0 & 0 & \left(\frac{1-\mu}{2}\right) \end{bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix}$$

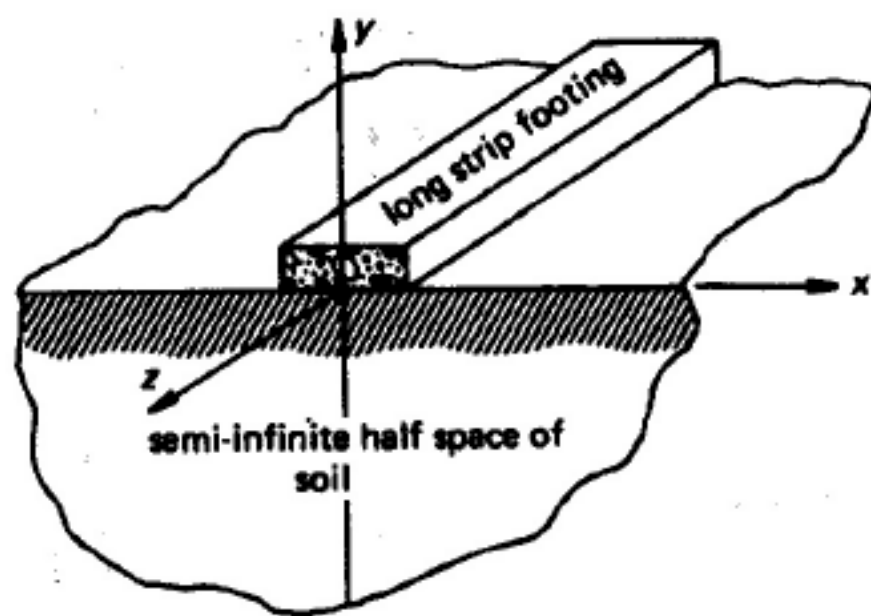
$$\text{or } \{\sigma\} = [D]\{\epsilon\}$$

Also strain in z – direction is

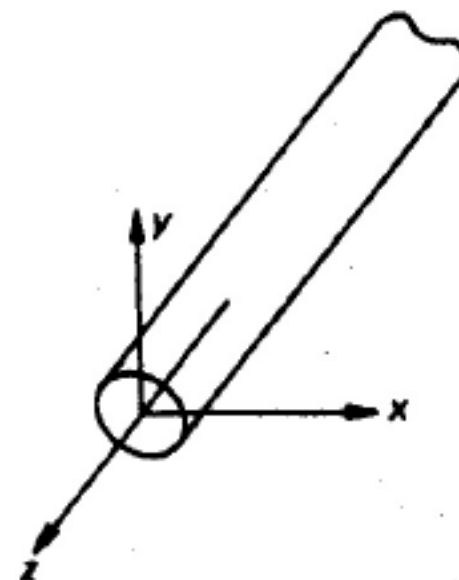
$$\epsilon_z = -\frac{\mu}{1-\mu}(\epsilon_x + \epsilon_y)$$

2. Plane Strain problem:

- Problems involving a long body whose geometry and loading do not vary significantly in the longitudinal direction (i.e. in z – direction) are referred as plain strain problems.
- Examples of plane strain problems are long strip footing, long cylinder and earth dam as shown in Figs. 7 (a) to 7(c).
- In these problems, the displacement component in z – direction is zero at every cross section and it is assumed that the displacements are the function of x and y coordinates only.



(a) Strip footing



(b) Long cylinder

Fig. 7: Plane strain problems

For plane strain problems, $\varepsilon_z = \gamma_{xz} = \gamma_{yz} = 0$ and the stress strain relation reduces to

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

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

Also stress in z – direction is given by

$$\sigma_z = \mu(\sigma_x + \sigma_y)$$

Introduction to Numerical Methods: **Rayleigh-Ritz method**

It is an approximate numerical technique in which the total potential energy π is used to obtain approximate solution of the continuum.

This method involves the construction of an assumed displacement field, say

$$\begin{aligned} u &= \sum a_i \phi_i(x, y, z) & i &= 1 \text{ to } l \\ v &= \sum a_j \phi_j(x, y, z) & j &= l+1 \text{ to } m \\ w &= \sum a_k \phi_k(x, y, z) & k &= m+1 \text{ to } n \end{aligned} \quad \text{---(a)}$$

The functions ϕ_i, ϕ_j, ϕ_k are usually taken as polynomials. a_i, a_j, a_k are constants which are to be determined. The displacement fields u, v, w must satisfy specified boundary conditions. The total potential energy of a 3D structure is expressed as

The total potential energy of a 3D structure is expressed as

$$\pi = \frac{1}{2} \iiint_V \{\sigma\}^T \{\epsilon\} dV - \iiint_V \{u\}^T \{f\} dV - \iint_S \{u\}^T \{T\} dS - \sum \{u_i\}^T \{P_i\} \quad \text{---(b)}$$

Now introducing stress-strain and strain-displacement relations and substitution of Eqn. (a) into above Eqn. (b) yields the total potential energy as a functions of constants a 's.

i.e. $\pi = \pi(a_1, a_1, \dots, a_r)$

Then, the extremum of π w.r.t a_i ($i = 1$ to r) yields the set of r equilibrium equations. $\frac{\partial \pi}{\partial a_i} = 0$

($i = 1, 2, \dots, r$)

The following steps are used to obtain the solution of a problem in Rayleigh-Ritze method:

1. Formulate potential energy functional for a given continuum (body).
2. Assume a trial displacement function for the given continuum. The function assumed should satisfy the boundary conditions.
3. Substitute the displacement function into potential energy functional.
4. Minimize the potential energy functional so as to get the equilibrium conditions.
5. Determine the unknown displacements and hence the stresses and strains.

Potential energy method

Principle of Minimum Potential energy

The principle of minimum potential energy states that *“for conservative system of all the kinematically admissible displacement fields, those corresponding to equilibrium extremize the total potential energy. If the extremum condition is a minimum, the equilibrium state is stable”*.

According to the principle of minimum potential energy for the system to be stable, the derivative of total potential energy w.r.t displacement ' u ' should be zero.

For a structure or deformable body, the total potential energy is the sum of the strain energy due to internal forces (i.e. due to stresses) and potential energy due to external forces (i.e. due to loads). Therefore,

$$\pi = U + W_p$$

where, U = strain energy and W_p = potential energy due to external forces or work potential.

(a) Strain energy (U):

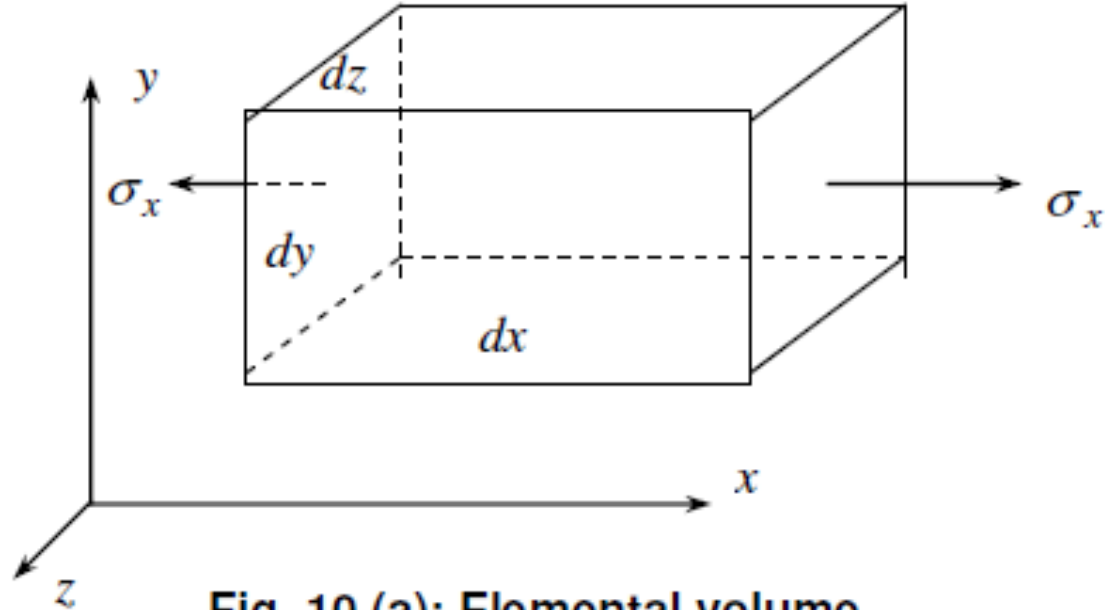


Fig. 10 (a): Elemental volume

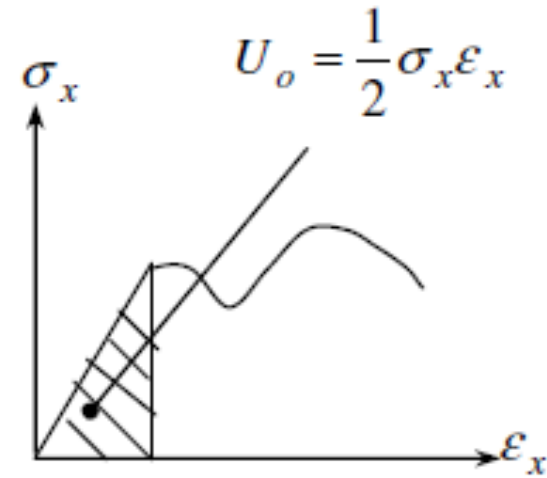


Fig. 10(b): stress vs strain

Consider an elemental volume dx , dy and dz subjected to stress ' σ_x ' only as shown in Fig. 10(a). Let ' ϵ_x ' be the strain due to this stress. Then the stress strain diagram is as shown in fig. 10(b).

Let U_o be the strain energy density (i.e. strain energy per unit volume). Then,

$$U_o = \frac{1}{2} \sigma_x \epsilon_x \text{ i.e. area of shaded portion in stress strain diagram.}$$

Extending the above expression for a 3 – D stress system, we get

$$U_o = \frac{1}{2} [\sigma_x \epsilon_x + \sigma_y \epsilon_y + \sigma_z \epsilon_z + \tau_{xy} \gamma_{xy} + \tau_{yz} \gamma_{yz} + \tau_{zx} \gamma_{zx}]$$

Or in matrix form,

$$U_o = \frac{1}{2} \{\sigma\}^T \{\epsilon\} \quad \text{---(a)}$$

where, $\{\sigma\}^T = [\sigma_x \quad \sigma_y \quad \sigma_z \quad \tau_{xy} \quad \tau_{yz} \quad \tau_{zx}]$ and $\{\epsilon\} = \{\epsilon_x \quad \epsilon_y \quad \epsilon_z \quad \gamma_{xy} \quad \gamma_{yz} \quad \gamma_{zx}\}$

The Eqn. (a) represents the strain energy per unit volume. The strain energy for the entire body is obtained by integrating the above equation over volume.

^

$$U = \iiint_V U_o dV = \frac{1}{2} \iiint_V \{\sigma\}^T \{\epsilon\} dV \quad \text{---(b)}$$

(b) Potential energy due to external forces or work potential (W_p):

The potential energy due to external forces (i.e. due to body & surface forces) is given by

$$W_p = - \iiint_V [f_x u + f_y v + f_z w] dV - \iint_S [T_x u + T_y v + T_z w] dS$$

where, u , v & w are the displacement components in x , y & z directions respectively. The above equation can be written in matrix form as

$$W_p = - \iiint_V \{u\}^T \{f\} dV - \iint_S \{u\}^T \{T\} dS$$

If the body is also subjected to concentrated load at point 'i', then the above equation becomes as

$$W_p = - \iiint_V \{u\}^T \{f\} dV - \iint_S \{u\}^T \{T\} dS - \sum \{u_i\}^T \{P_i\} \quad \text{---(c)}$$

where, $\{u_i\}^T = \{u_i \quad v_i \quad w_i\}$ is the displacement components corresponding to point 'i' where concentrated load ' P_i ' is acting.

(c) Total potential energy (π):

The total potential energy is given by $\pi = U + W_p$

Substituting from Eqns. (b) and (c) we get

$$\pi = \frac{1}{2} \iiint_V \{\sigma\}^T \{\epsilon\} dV - \iiint_V \{u\}^T \{f\} dV - \iint_S \{u\}^T \{T\} dS - \sum \{u_i\}^T \{P_i\}$$



A T M E
College of Engineering





A T M E
College of Engineering





A T M E
College of Engineering





THANK YOU

Difference between Finite Element Method and Continuum method.

Finite Element Method	Continuum method
1. The results obtained are realistic and very near to exact values.	1. Gives an approximate solution.
2. Most of the practical engineering problems can be solved.	2. Difficult to obtain solutions for practical problems.
3. Geometric and material non-linearities can be handled effectively.	3. Geometric and material non-linearities cannot be handled easily.
4. Solutions are obtained on an elemental basis	4. Solutions obtained for a whole domain problem.
5. Solution to complex geometry can be obtained	5. Solutions to simple and regular shape can be obtained.