

FINITE ELEMENT METHOD

1. INTRODUCTION

FEM is a numerical technique, which can be used for any field of engineering problems such as stress analysis, fluid flow, heat and mass transfer, lubrication, combustion, electromagnetic etc. With the advances in computer technology any kind of complex problems can be made easy using FEM.

In FEM, a complex region defining a continuum or solution domain is divided into simple geometric shapes called *finite elements* as shown in Fig. 1. These elements are interconnected at joints, which are called *nodes*. The material property and governing equations are considered over these elements and expressed in terms of unknown variables at element nodes. Then, an assembly process results into a set of simultaneous equations. The solution of these simultaneous equations gives the approximate behavior of the solution domain.

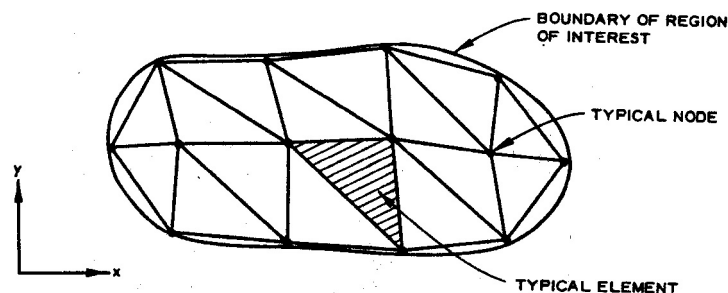


Fig. 1: Two-dimensional region divided into finite element

2. FEM Vs Classical Methods

1. In classical method exact equations are formed and exact solutions are obtained whereas in FEM exact equations are formed but approximate solutions are obtained.
2. Solutions can be obtained only for few simple cases by classical method whereas it can be obtained for all problems by FEM.
3. Whenever the complexities in shape, material, boundary conditions and loading as shown in Fig.2 are occurs in the problems, classical method provides the solution with drastic assumptions whereas in FEM no such assumptions are made.
4. The solutions for the problems with anisotropic material properties become very difficulty in classical method. FEM can handle problems with anisotropic material properties without any difficulty.
5. If the structure consists of more than one material, it is difficulty to use classical method but FEM can be used with out such difficulty.
6. Problems with material and geometric non-linearities cannot be handled by classical method but there is no such difficulty in FEM.

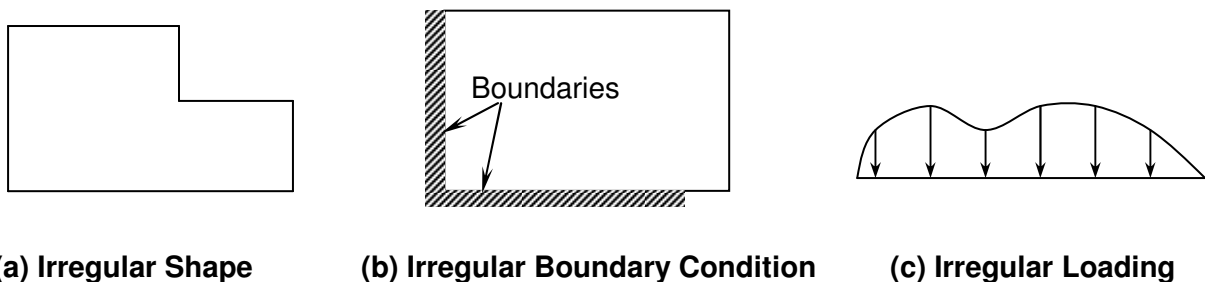


Fig. 2

3. Finite Difference Method (FDM)

In finite difference method, the body or the system is discretized by a mesh of nodal points as shown in Fig. 3. The field variable is represented by the discrete values of the variable at the nodes. For example, in case of plate bending problem the normal displacement w will be taken as unknown variable at each one of the nodes. The governing differential equation and the boundary conditions are converted to finite difference form.

The finite difference form of governing differential equation and boundary conditions are then applied to each of nodes. This will yield a set of linear simultaneous equations in the discrete field variable w . The resulting simultaneous equations are solved for the nodal values of this variable.

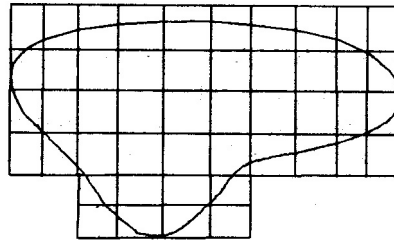


Fig. 3: Discretization of region by mesh for finite difference method

4. FEM Vs FDM

1. FDM makes point wise approximation to the governing equations i.e. it ensures continuity only at the node points but continuity along the sides of grid lines are not ensured.
FEM makes piece wise approximation i.e. it ensures continuity at node point as well as along the sides of the element.
2. FDM does not give the values for the unknown variables at any point except at node points. It does not give any approximating function to evaluate the basic values (deflection in case of structural problems or temperature in case of heat transfer problems) using the nodal values.
FEM provides values for the unknown variables at node points and nodal variables can be interpolated using suitable function to get the values at any point other than nodes.
3. FDM makes stair type approximation to irregular boundaries as shown in Fig.3 whereas in FEM irregular boundary can be discretized exactly using suitable elements.
4. FDM needs more number of nodes to get good results whereas FEM needs less number of nodes.
5. Complicated problems such as irregular shapes, material properties, boundary conditions and loading can't be handled by FDM but these problems can be handled by FEM.

5. Advantages and Disadvantages of FEM over other Numerical Techniques such as Finite Difference Method (FDM):

Advantages

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

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.

6. Applications of FEM

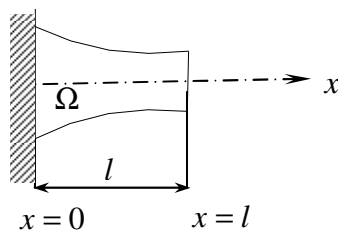
The applications of FEM, in general, ranges from deformation and stress analysis of automotive, aircraft, building and bridge structures to field analysis such as heat flux, fluid flow, magnetic flux, seepage and other flow problems. Some examples of specific applications of FEM for the important boundary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

7. Some Definitions

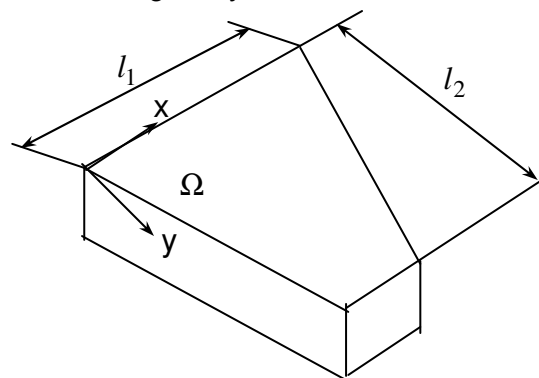
1. Continuum or Solution domain (Ω):

Continuum is the physical body, structure or solid being analysed.



$$\Omega \Rightarrow 0 \leq x \leq l$$

Fig. (a): 1-D solution domain



$$\begin{aligned} \Omega &\Rightarrow 0 \leq x \leq l_1 \\ &\Rightarrow 0 \leq y \leq l_2 \end{aligned}$$

Fig. (b): 2-D solution domain

2. **Field variable:** Field variable is any variable such as displacement, pressure, temperature, velocity component etc.

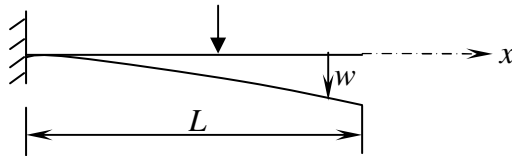
3. Nodal variables: The values of field variables at nodes are called *nodal variables*. Generally, the nodal variables are the unknown of the problem and hence the analysis of FEM concerned with finding nodal variables.

4. Boundary conditions: Boundary conditions are the specified values of the field variables (or their derivatives) on the boundaries of the problems. There are two types of boundary conditions. They are (1) Essential or geometric or kinematic boundary conditions and (2) Non-essential or natural or forced boundary conditions.

(1) *Essential boundary conditions:* The essential boundary conditions are the prescribed values of the variables specified (or their first derivative) on the boundary of the region.

For example, if a standard cantilever beam shown in Fig. is used, the deflection, w and slope

$\frac{dw}{dx}$ is zero at $x = 0$ (i.e at fixed point). These boundary conditions, $w = 0$ and $\frac{dw}{dx} = 0$ are known as essential boundary conditions.



(2) *Non-Essential boundary conditions:* Non-essential boundary conditions are the prescribed values of higher derivatives of the field variables.

For example, in the cantilever beam shown, the non-essential boundary conditions are its bending moment and shear force at its free length (i.e. at $x = L$) and are expressed as

$$\text{At } x = L, \text{ B.M., } EI \frac{d^2 w}{dx^2} = 0 \text{ and shear force } EI \frac{d^3 w}{dx^3} = 0.$$

5. Boundary value problem:

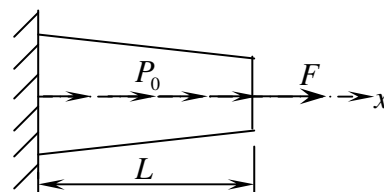
Boundary value problem is one in which a solution is obtained in the region of the body while on the boundaries (or edges) of the region the values of the dependent variables (or their derivatives) are prescribed. Boundary value problems are also some times called *field problems*. *structural problems, soil mechanics problems, thermal problems, etc are the boundary value problems.*

As an example consider a structural problem shown in Fig. Here 1D bar is subjected to a uniformly distributed load, P_0 along its length and a concentrated load, F at its free end. The governing differential equation for this problem can be expressed as

$$\frac{d}{dx} \left(AE \frac{du}{dx} \right) - P_0 = 0 \text{ for } 0 < x < L \quad \text{---(a)}$$

and the boundary conditions are

$$u = 0 \text{ at } x = 0 \text{ and } AE \frac{du}{dx} = F \text{ at } x = L \quad \text{--- (b)}$$



The problems which are represented by a governing equation such as Eq. (a) and boundary conditions given by Eq. (b) are referred as boundary value problems.

6. Initial value problem:

Initial value problem is one in which the dependent variables (or their derivatives) are specified initially (i.e. at time $t = 0$). Initial value problems are generally time dependent problems. Vibration analysis of a body is an example for the initial value problem. As an example the governing differential equation of a spring-mass system subjected to excitation force, $F(t)$ is expressed as

$$m \frac{d^2 u}{dt^2} + ku = F(t) \quad \text{--- (a)}$$

with initial condition

$$u = u_0 \text{ and } \frac{du}{dt} = v_0 \text{ when, time } t = 0. \quad \text{--- (b)}$$

i.e the displacement of mass from its equilibrium position and its velocity when time $t = 0$ are u_0 and v_0 respectively. Those problems represented by such a governing Eq. (a) and such initial conditions shown in eq. (b) are referred as initial value problems.

7. Degrees of freedom per node: It is the number of nodal variables required to determine per node.

8. Degrees of freedom per element: It is the number of nodal variables required to determine per element.

Let n_{dn} = No. of d.o.f per node

n_{de} = No. of d.o.f per element.

n_e = No. of nodes per element.

Then, $n_{de} = n_e * n_{dn}$

(1) For 1- D problem,

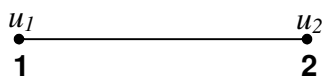


Fig. 4(a): 2-noded bar element

1 – D.O.F per node

2 – D.O.F per element

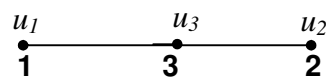


Fig.4(b): 3-noded bar element

1 – D.O.F per node

3 – D.O.F per element

(2) For 2- D problem,

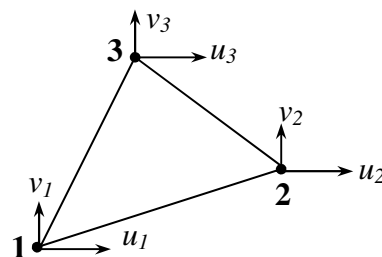


Fig.4(c): 3 – noded triangular element

2 – D.O.F per node

6 – D.O.F per element

9. Equilibrium equation of an elastic body:

The equilibrium equation of an elastic body relates the internal forces with that of external forces under the equilibrium of the body. Equilibrium equation is usually derived from the principle of minimum potential energy.

The total potential energy of a loaded elastic body is given by the sum of the internal energy stored as result of deformation and potential energy of the external load.

Consider a column loaded as shown in Fig. 5(a) and having deformation u . This column can also be by a linear spring of stiffness K as shown in Fig. 5(b). The spring force v/s deformation is as shown in Fig. 5(c).

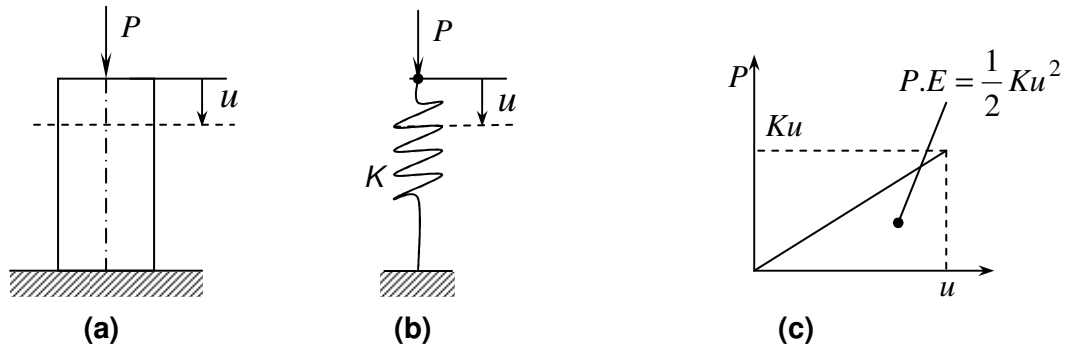


Fig. 5: Column and equivalent spring

Total P.E, $\pi = U + W_p$

Where, $U = \frac{1}{2} Ku^2$, Strain energy or internal potential energy

And $W_p = -Pu$, Potential energy due to external load

Note that negative sign in the above equation is because the force loses some of the capacity for doing work when it is displaced in the direction it acts.

Therefore, total P.E, $\pi = \frac{1}{2} Ku^2 - Pu$ --- (a)

For the body to be in state of equilibrium, the total P.E is to be minimum.

Therefore, for total P.E to be minimum, $\frac{\partial \pi}{\partial u} = 0$

Differentiation of Eqn. (a) w.r.t u yields,

$$Ku - P = 0$$

or $Ku = P$ is the equilibrium equation.

Where, Ku = Internal force exerted by the spring and P = External force.

If both end of the spring or body is having displacement as shown in Fig. 4(d), the equilibrium equation becomes, in matrix form, as

$$\begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix}$$

In general, the above equation for an e^{th} body or element having nodes i and j can be written as

$$\begin{bmatrix} K_{ii} & K_{ij} \\ K_{ji} & K_{jj} \end{bmatrix}^e \begin{Bmatrix} u_i \\ u_j \end{Bmatrix}^e = \begin{Bmatrix} P_i \\ P_j \end{Bmatrix}^e \quad \text{--- (b)}$$

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

where,

$[K]^e$ = Element stiffness matrix

$\{q\}^e$ = Element nodal displacement vector and

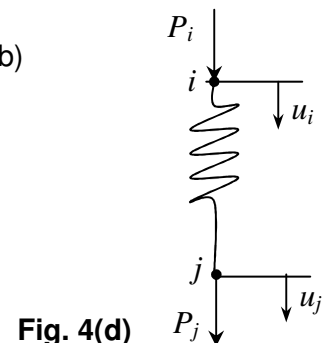


Fig. 4(d)

$\{F\}^e$ = Element nodal force vector.

The elements of stiffness matrix in Eqn. (b) are called as influence coefficients. The influence coefficient ' K_{ij} ' is defined as the force required at node i to hold that node fixed when a unite displacement at node j is given. On the other hand, the coefficient ' K_{ji} ' is the force required at node ' j ' to give unite displacement at same node ' i '. These coefficients can be expressed in terms of area of c/s of the elastic body, length and material property i.e. Young's modulus of the body and are given by

$$K_{ii} = K_{jj} = \frac{A_e E_e}{l_e} \text{ and } K_{ij} = K_{ji} = -\frac{A_e E_e}{l_e}$$

Then, Eqn. (b) becomes as

$$\frac{A_e E_e}{l_e} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix}^e = \begin{Bmatrix} P_i \\ P_j \end{Bmatrix}^e, \text{ is the equilibrium equation for an } e^{\text{th}} \text{ element.}$$

8. Finite Element Method Approaches

Majority of the application of FEM is in the area of solid mechanics and hence the problems in these fields are usually tackled by one of the following three approaches:

1. Displacement method,
2. Equilibrium or Flexibility method and
3. Mixed method.

Displacements are assumed over the element as primary unknown quantities in displacement method; stresses are assumed over the element as primary unknown quantities in equilibrium method and some displacements and some stresses are assumed as primary unknown quantities in the mixed method.

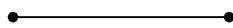
9. Steps Involved in FEM

The following are basic steps used in FEM. They are

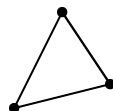
1. Discretization of the continuum: Discretization is the process of subdividing a given continuum into finite elements as shown in Fig. 1. The finite elements may be triangular or quadrilateral etc. for a two-dimensional problems; they may be tetrahedron or rectangular prism etc. for three-dimensional problems. The selection of finite elements in FEM is depends on the following factors. They are:

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.

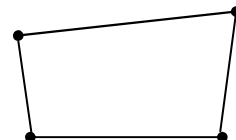
Fig. 7 shows some of typical finite elements used in FEM.



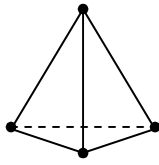
(a) Line element
for 1 – D problem



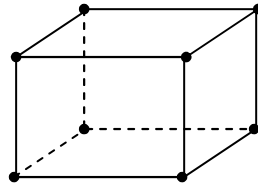
(b) 3-noded triangle
for 2 – D problem



(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. 7: 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 $l_1 = l_2 = 1\text{m}$; $A_1 = 20\text{m}^2$, $A_2 = 10\text{m}^2$ and $E_1 = E_2 = 2 \times 10^{11} \text{ N/m}^2$.

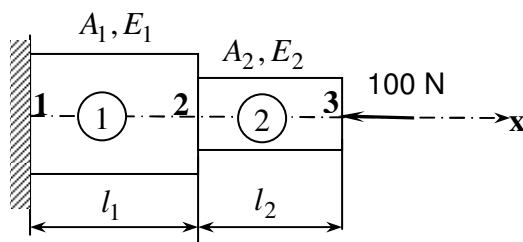


Fig. 8 (a): Stepped column

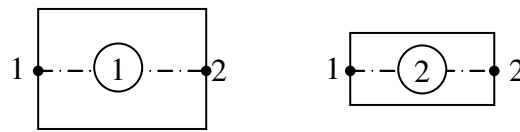


Fig. 8(b): Discretization

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,

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

$$\{q\}^e = \begin{Bmatrix} u_i^e & u_j^e \end{Bmatrix}^T, \text{ nodal displacement vector and}$$

$$\{F\}^e = \begin{Bmatrix} P_i^e & P_j^e \end{Bmatrix}^T, \text{ nodal load vector.}$$

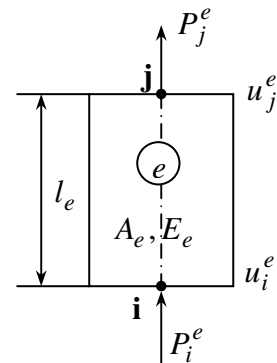


Fig. 9: Typical element

The element stiffness matrices for the example problem considered are:

For element-1: $\frac{A_1 E_1}{l_1} = \frac{20 * 2 * 10^{11}}{1} = 4 * 10^{12} \text{ N/m}$

Therefore, $K^1 = 10^{12} \begin{bmatrix} 1 & 2 \\ 4 & -4 \\ -4 & 4 \end{bmatrix} \begin{matrix} 1 \\ 2 \end{matrix}$

For element-2: $\frac{A_2 E_2}{l_2} = \frac{10 * 2 * 10^{11}}{1} = 2 * 10^{12} \text{ N/m}$

Therefore, $K^2 = 10^{12} \begin{bmatrix} 2 & 3 \\ 2 & -2 \\ -2 & 2 \end{bmatrix} \begin{matrix} 2 \\ 3 \end{matrix}$

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

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

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

$$\{F\} = \begin{Bmatrix} P_1 \\ P_2 \\ P_3 \end{Bmatrix} = \begin{Bmatrix} R \\ 0 \\ -100 \end{Bmatrix} \begin{matrix} \text{Reaction at fixed node 1} \\ \text{As there is no external load at node 2} \\ \text{As external load at node 3 acts in - ve x - direction} \end{matrix}$$

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 Eqs. we get $u_3 = -7.5 * 10^{-12}$ m.

Note: The reaction at fixed node 1 is obtained by substituting the values of nodal displacements u_1, u_2, u_3 into 1st equation of Eqn. (A).

$$\text{i.e. } 4 * 10^{12} u_1 - 4 * 10^{12} u_2 = R$$

$$\text{or } R = -4 * 10^{12} (-25 * 10^{-12}) = +100 (\uparrow) \quad \text{since } u_1 = 0$$

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

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

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

$$\text{For element 1: Strain, } \epsilon_1 = \frac{u_2 - u_1}{l_1} = \frac{-25 * 10^{-12} - 0}{1} = -25 * 10^{-12} \text{ (compressive)}$$

$$\text{Stress, } \sigma_1 = E_1 * \epsilon_1 = 2 * 10^{11} * -25 * 10^{-12} = -5 \text{ N/m}^2$$

$$\text{For element 2: Strain, } \epsilon_2 = \frac{u_3 - u_2}{l_2} = \frac{-7.5 * 10^{-12} - (-25 * 10^{-12})}{1} = -50 * 10^{-12} \text{ (compressive)}$$

$$\text{Stress, } \sigma_2 = E_2 * \epsilon_2 = 2 * 10^{11} * -50 * 10^{-12} = -10 \text{ N/m}^2$$