CHAPTER 4

AN INTRODUCTION OF THE FINITE ELEMENT METHOD

4-1 Definition:

The finite element method is a tool to solve one dimensional, two – dimensional and three – dimensional structures with approximation instead of solving complicated partial differential equations. The structure is discredited into a set of elements joined together at some points called nodes or nodal points. These nodes are similar to the joints in the one – dimensional structures which were investigated in the previous chapters. The nodes could be the common corners between the elements, or chosen between the boundaries of the elements. The similarity in the concept between one – dimensional skeleton structures and two or three – dimensional structures, in terms of discretization. Engineering-FEM is a procedure for determining the Deformation and/or stress field in a structural system. The global structural response is determined by modeling the relatively simple response of pieces of the system and combining the response of these individual components to determine
It is obvious that in one-dimensional structures the element is a one-dimensional member. In two-dimensional structures, the element is a two-dimensional plate or shell element. The three-dimensional element could be a cube, prism, or a tetrahedron, either with straight sides.

The solution of the finite element method is almost the same as the direct stiffness matrix method. Once the elements stiffness matrices are found, these matrices are augmented according to the compatibility and equilibrium conditions at every node. The free nodal displacements can be determined after specifying the boundary conditions at the boundary nodes. However, what interests the analysis in two-dimensional and three-dimensional structures is the stresses and strains not forces and displacements. Therefore, it is
necessary to relate the strains at a point within the element with the nodal displacements.

4-2 Course objectives:

Students will have the Knowledge and skills to use the finite element method to predict stress and strain fields is elastic structural subassemblies subjected to a variety of static load conditions.

1- Understand the theory of the finite element method and demonstrate this understanding by formulating the finite element problem.

2- Solve a global structural analysis problem for a structure and solution.

3- Use a typical finite element analysis software package to analyze structures and interpret the results of these analyses.

4-3 Mathematical view of the FEM:

It is a numerical method that provides an approximate solution to a boundary value problem that is defined by a partial or ordinary differential equation.

4-4 Engineering View of the FEM:

It is a method of determining the global response of a complex structure by modeling the relatively simple response of small pieces of system and combining these pieces to determine the global response
Element with constant area, elastic modulus and stress. We know the relationship between stress, strain and force for these simple elements.

4-5 Why use F.E.M?

1-Because analytical solution can be found only for very simple problems:

\[ V \cdot \sigma = 0 \text{ for } x \in \Omega \]
\[ \sigma_{xx} = -S \text{ for } x \in \Gamma_{g1} \]
\[ \sigma_{xx} = S \text{ for } x \in \Gamma_{g2} \]
\[ \sigma_{zz} = \sigma_{xz} = \sigma_{yz} = 0 \text{ for } x \in \Omega \]
2- It provides a tool for use in determining the response of complex structures.

\[ \tau_{rr} = \frac{S}{2} \left[ \left( 1 - \frac{a^2}{r^2} \right) + \left( -1 + 4 \frac{a^2}{r^2} - 3 \frac{a^4}{r^4} \right) \cos \theta \right] \]

\[ \tau_{\theta\theta} = \frac{S}{2} \left[ \left( 1 - \frac{a^2}{r^2} \right) + \left( 1 + 3 \frac{a^4}{r^4} \right) \cos 2\theta \right] \]

3- Complex structures in the design office, Structural wall with irregular openings subjected to simulated earthquake loading
4- Slab with irregular openings subjected to gravity loading.

5- Complex structures in the design office diaphragms.
6- It enables high-resolution prediction of (stress /strain /damage) Fields

Cypress viaduct, Oakland CA Following loma prieta Earthquake in 1989

7- The mathematical formulation of the F.E.M. provides:

* Proof of convergence.
* Methods for estimating errors.

8- Formulation provides a means of enhancing individual aspects of the model.

**4-6 Steps in the FEM:**

**4-6-1 Assumptions:**

1-we are solving a structural engineering problem.
2-The unknowns we seek are nodal displacements that define the displacement along the edges of the pieces that compose the body. From these we can compute strains and stresses.

3-The solution is a displacement field that satisfies equilibrium, compatibility and constitutive requirements.

**Step 1: Discrete the body**

1-Element size (fidelity of the field, accuracy, computational effort).
2-Type of element (behavioral mechanisms that are represented).

<table>
<thead>
<tr>
<th>Wall frame systems</th>
<th>Frame elements</th>
<th>Plane stress or shell elements</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Wall frame system" /></td>
<td><img src="image2" alt="Frame elements" /></td>
<td><img src="image3" alt="Plane stress or shell elements" /></td>
</tr>
</tbody>
</table>

**Steps2: select a displacement function within each element**

1- The displacement function is associated with the element type (beam elements are assumed to exhibit flexural deformation but not shear deformation, plane strain elements are assumed to deform only in plane)

2- Displacement function is usually very simple polynomials.
3- Typically all elements of one type are assumed to have the same displacement function.

4- Displacement within the element is defined by the displacement function + the nodal displacements.

**4-6-2 Element Formulations & Displacement Fields:**
1-DTruss Element (1 displacement D.O.F. per node in direction of bar axis. Two nodal displacements enable definition of linear displacement function).

2-D Frame element (1 displacements and 1 rotational D.O.F. per node in the direction perpendicular to the axis of the element. Four nodal (displacement) enable definition of a cubic displacement field).

2-D plan stress and plane strain Elements (2 orthogonal displacement D.O.F. per node, displacement is linear in x&y with one cross term).

Pseudo 3-D Axisymmetric Elements (2 orthogonal disp. D.O.F. per node, within a plane displacement is linear in z&r with one cross term).
Plate element (2 rotations and 1 out of – plan disp. D.O.F. per node. the out of plan displacement field is cubic in x&y with cross terms).
Shell Elements (3 displacement and 2 rotation D.O.F. per node).

Out of plan bending + In-plan (membrane) action

**Step 3:** Define the strain displacement (kinematics) and stress strain relationships (constitution).

**Step 4:** Derive the element stiffness matrix and equilibrium equations using direct equilibrium method _this is probably what you have done so far. Or Work & Energy Method _ the principle of minimum potential energy. Castiglianos theorem & the principle of virtual work and method of weighted residual _ Galerkins method.

**Step 5:** Assemble the element equilibrium equations to obtain the global equilibrium equations and introduce boundary conditions.

\[ F = KD \]

**Step 6:** Solve for unknown nodal displacements.

**step 7:** Solve for element stresses and strains.
**Step 8:** Interpret the results.

### 4-7 Types of elements:

1. Beam Elements. (BE)
2. Triangle Elements. (ΔE)
3. Four Node Elements. (4NE)
4. Rectangle Elements. (RE)
5. Four Node Quadratic Elements. (4NQE)
6. Eight Node Quadratic Elements. (8NQE)
7. Four Node Quadratic Isoperimetric Elements. (4NQIE)
8. Friction Element. (FE)
9. Fracture Element. (CE)
10. Infinite Elements. (IE)

**Example: Construct the Finite Element Mesh**

![Finite Element Mesh Example]

**4-8 The Unit Displacement Method:**

The Unit Displacement Method (UDM) is used for computing the stiffness matrix for truss and beam finite element models.
This method will be demonstrated first using a simple 2-node, 1-D truss element and later will be used to produce the stiffness matrix for a 2-node, 2D truss element.

Given the 2-node, 1-D truss element shown below:

We know the element stiffness equation can be written as:

\[
\begin{bmatrix}
  k^e & -k^e \\
  -k^e & k^e 
\end{bmatrix}
\begin{bmatrix}
  u_i \\
  u_{i+1}
\end{bmatrix} =
\begin{bmatrix}
  F^e_{i} \\
  F^e_{i+1}
\end{bmatrix}
\]

But let's put this equation in more generic form where \(k_{11}, k_{12}, k_{21}, \) and \(k_{22}\) are unknown stiffness coefficients.

\[
\begin{bmatrix}
  k_{11} & k_{12} \\
  k_{21} & k_{22}
\end{bmatrix}
\begin{bmatrix}
  u_1 \\
  u_2
\end{bmatrix} =
\begin{bmatrix}
  F_1 \\
  F_2
\end{bmatrix}
\]

We can use the unit displacement method to solve for the stiffness in the generic form of the element stiffness equation.

In the Unit Displacement Method, we define a single nodal displacement to be equal to 1 and all other nodal displacements to be equal to zero. Using this definition we calculate corresponding forces.
IMPORTANT NOTE:
A unit displacement (=1) is assumed to be a very small displacement which does not significantly change the geometry of the element.

For the previous example, using the Unit Displacement Method, we get:

\[
\begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix}
\]

\[
\begin{align*}
u_1 &= 1 & k_{11} \cdot 1 + k_{12} \cdot 0 &= F_1 & \kappa_{11} &= F_1 \\
u_2 &= 0 & k_{21} \cdot 1 + k_{22} \cdot 0 &= F_2 & \kappa_{21} &= F_2
\end{align*}
\]

Now use our Mechanics of Material equations assuming equilibrium around the element where \( u_1 = 1 \) and \( u_2 = 0 \), we have:

\[
\begin{align*}
k \cdot (u_1 - u_2) &= F_1 & k \cdot (1 - 0) &= F_1 \\
k \cdot (u_2 - u_1) &= F_2 & k \cdot (0 - 1) &= F_2
\end{align*}
\]

Thus:

\[
\begin{align*}
F_1 &= k \\
F_2 &= -k \\
\kappa_{11} &= k \\
\kappa_{21} &= -k
\end{align*}
\]
Using the same logic but setting $u_1=0$ and $u_2=1$:

$$F_1 = k_{12} = -k$$ and $$F_2 = k_{22} = k$$

Thus:

$$\begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix}$$

Repeat this method to determine the stiffness matrix for the 2-node, 2-D truss element below:

Note, that a node could only move horizontally in the 1D truss element, however, a node in a 2D truss element can move in the x- or y-direction. We therefore say that the 1D truss element has 1 degree of freedom (1DOF) per node while the 2D truss element has 2DOF’s per node.

The four nodal equilibrium equations for the element can be written in matrix form as:
Where $k_{11}$, ..., $k_{44}$ are sixteen unknown stiffness coefficients to be determined.

First examine the case where:

$U_{1x}=1$
$U_{2x}=0$
$U_{1y}=0$
$U_{2y}=0$

Performing matrix math, the nodal forces are given by $F_{1x}=k_{11}$, $F_{1y}=k_{21}$, $F_{2x}=k_{31}$, $F_{2y}=k_{41}$

Since node 2 is fixed and node 1 has moved a distance of 1 in the x-direction, the element length has shortened a distance, $\cos(\theta)$.
Thus the force generated in the element is: $F_e = k_e \delta$

Note, $d = \cos(\theta)$, therefore $F_e = k_e \cos(\theta)$

Therefore, $F_e = k \cos(\theta)$ when the element stiffness is $k$. Furthermore, geometrically we see that:

$$F_{1x} = F_e \cos(\theta)$$
$$F_{1y} = F_e \sin(\theta)$$

Have equations from three different methods Matrix Manipulation Trigonometry Mechanics of Materials

$$F_{1x} = k_{11} \quad F_{1x} = F_e \cos(\theta)$$
$$F_{1y} = k_{21} \quad F_{1y} = F_e \sin(\theta)$$

Spring Law

Substitute and solve:

$$F_{1x} = k_{11} = k_e \cos^2(\theta)$$
$$F_{1y} = k_{21} = k_e \cos(\theta) \sin(\theta)$$

The force components at node 2 are equal and opposite to the force components at node 1, thus:

$$F_{1x} = k_{11} = k_e \cos^2(\theta)$$
$$F_{1y} = k_{21} = k_e \cos(\theta) \sin(\theta)$$

at node 2:
\[ F_{2x} = k_{31} = -k_e \cos^2(\theta) \]
\[ F_{2y} = k_{41} = -k_e \cos(\theta) \sin(\theta) \]

Repeating this procedure for the three other unit displacement combinations, we obtain all the elements of the stiffness matrix.

\[
[K] = \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} \\ k_{21} & k_{22} & k_{23} & k_{24} \\ k_{31} & k_{32} & k_{33} & k_{34} \\ k_{41} & k_{42} & k_{43} & k_{44} \end{bmatrix} = k_e \begin{bmatrix} \cos^2(\theta) & \cos(\theta) \sin(\theta) & -\cos^2(\theta) & -\cos(\theta) \sin(\theta) \\ \cos(\theta) \sin(\theta) & \sin^2(\theta) & -\cos(\theta) \sin(\theta) & -\sin^2(\theta) \\ -\cos^2(\theta) & -\cos(\theta) \sin(\theta) & \cos^2(\theta) & \cos(\theta) \sin(\theta) \\ -\cos(\theta) \sin(\theta) & -\sin^2(\theta) & \cos(\theta) \sin(\theta) & \sin^2(\theta) \end{bmatrix}
\]

**Note,** that this element stiffness matrix depends on the material properties, x-section area, length and angle of inclination with respect to the global horizontal direction.

### 4-9 2D Element Stiffness Matrix:

Comparing the 2-D element stiffness matrix with the 1-D truss stiffness matrix, a pattern emerges:

2D element \( \Leftrightarrow \) 1D element

2 x 2 color block <=equivalent=> 1 x 1 color block

\[
[K] = \begin{bmatrix} k_e \begin{bmatrix} CC & CS \\ CS & SS \end{bmatrix} & -k_e \begin{bmatrix} CC & CS \\ CS & SS \end{bmatrix} \\ -k_e \begin{bmatrix} CC & CS \\ CS & SS \end{bmatrix} & k_e \begin{bmatrix} CC & CS \\ CS & SS \end{bmatrix} \end{bmatrix} = \begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix}
\]

Where: \( k_e = \frac{EA}{L}, \) \( CC = \cos^2(\theta), \) \( CS = \cos(\theta) \sin(\theta), \) \( SS = \sin^2(\theta) \)
Therefore, the 2D element stiffness matrix is comprised of one 2 x 2 block matrix \([k_b]\) arranged so that:

\[
[K] = \begin{bmatrix}
[k_b] & -[k_b] \\
-[k_b] & [k_b]
\end{bmatrix}
\]

where \(CC = \cos^2(\theta), \ CS = \cos(\theta) \times \sin(\theta), \ SS = \sin^2(\theta)\) and the 2 x 2 block matrix \([k_b]\) is defined by:

\[
[k_b] = k_e \begin{bmatrix}
\cos^2(\theta) & \cos(\theta) \times \sin(\theta) \\
\cos(\theta) \times \sin(\theta) & \sin^2(\theta)
\end{bmatrix}
\]

4-10 3D Element Stiffness Matrix:

For a 3D truss element, the element stiffness matrix the mathematical structure of \([K]\) is the same as for the 2D element:

\[
[K] = \begin{bmatrix}
[k_b] & -[k_b] \\
-[k_b] & [k_b]
\end{bmatrix}
\]

Where \([k_b]\) is a 3x3 block matrix and \([K]\) is a 6 x 6 element stiffness matrix.

4-11 Two Dimensional Problems:

Two-dimensional problems in structural mechanics occur in a variety of circumstances. The most commonly encountered two-dimensional problem is that of thin plate subjected to in-plane edge loads.
4-11-1 Two Dimensional Elements:

Two-dimensional problems are typically modeled using triangular or quadrilateral elements.

4-12 Beam Element:

Beam elements are commonly used in the modeling of skeletal structures. One may use either plane or space elements according to the structure to be analyzed. Only plane element is introduced in this thesis, as in our analysis we are encountered by plane problems in which the element models the conduit walls.

The plane beam element has 2 nodes, one at each of its ends. At each node, 3 degrees of freedom exist. The degrees of freedom are horizontal displacement, vertical displacement and rotation from horizontal to vertical direction. The element have six degrees of freedom consequently, the element stiffness matrix is \((6 \times 6)\) matrix.
The formulation of stiffness matrix develops from the physical meaning of the matrix elements. Let $k_{ij}$ be the value of the element in row no. $i$ and column no. $j$. This value represents the force induced in the direction of freedom no. $i$ due to a unit displacement in the direction of degree of freedom no. $j$ while the other degree of freedom are restricted. Hence, the end forces resulting from the previous displacement configuration are the values of column $j$ in the stiffness matrix.

Let us consider a beam element, in a general position, inclined to the X-axis by an angle $\theta$ (positively measured in the counter-clockwise sense). Also let the properties of the element material are:

**E:** Young's modulus of the element material.

**I:** second moment of area of the cross-section about the axis perpendicular to the plane of element and passes through the centroid of the cross-section.

**A:** Cross sectional area.

Furthermore, the coordinates of the element nodes are $(X_1, Y_1)$ and $(X_2, Y_2)$. Then the projections of the element along X and Y axes are $X_p$ and $Y_p$ respectively.

\[
\begin{align*}
X_p &= X_2 - X_1 \\
Y_p &= Y_2 - Y_1
\end{align*}
\]

The element length
\[
L = \sqrt{[(X_p)^2 + (Y_p)^2]}
\]

Now a unit displacement (positive) is given to the degree of freedom, $q_1$ whereas degrees $q_2$ and $q_6$ restricted.
The resulting end forces constitute the first column in the stiffness matrix.
The displacement $q_1 = +1$ is resolved into two components parallel and perpendicular to element direction, the forces due to each component are calculated in the direction of the displacement components.

Plane Beam Element

Plotting of Beam Element in Cartesian Coordinates
The forces due to the two components are added together. Then resolved in the directions of the original degrees of freedom.

Hence, the first column is:

\[
\begin{bmatrix}
\frac{EA \cos^2 \theta}{L} & \frac{12 EI \sin^2 \theta}{L^3} \\
\frac{EA \sin \theta \cos \theta}{L} & - \frac{12 EI \sin \theta \cos \theta}{L^3} \\
- \frac{6 EI \sin \theta}{L^2} & - \frac{EA \cos^2 \theta}{L} + \frac{12 EI \sin^2 \theta}{L^3} \\
- \frac{EA \sin \theta \cos \theta}{L} & + \frac{12 EI \sin \theta \cos \theta}{L^3} \\
- \frac{6 EI \sin \theta}{L^2} & \frac{L^2}{2}
\end{bmatrix}
\]

Similarly, unit displacement is given to the degrees q2, q3, q4, q5, q6 one by one and keeping the other degrees restricted.

**The end forces can be related to end displacements as follows:**

\[
\begin{align*}
N_1 &= (D_4 - D_1) * E * A/L = E * A/L \ (D_4 - D_1) \\
N_2 &= N_1 \\
Q_1 &= 12 E * I / L^3 (D_2 - D_5) + 6 E * I / L^2 (D_6 - D_3) \\
Q_1 &= Q_2 \\
M_1 &= 6 E * I / L^2 (D_5 - D_2) - 2 E * I / L (D_6 + 2D_3) \\
M_2 &= 6 E * I / L^2 (D_2 - D_5) - 2 E * I / L (D_3 + 2D_6)
\end{align*}
\]

\[
[K] =
\begin{bmatrix}
\frac{c_1 x_p^2 + c_2 y_p^2}{2} & \frac{c_1 x_p y_p + c_2 x_p y_p}{2} & -\frac{c_2 y_p^2}{2} & -\frac{c_2 y_p^2}{2} \\
-\frac{c_1 x_p^2}{2} & -\frac{c_1 x_p y_p - c_2 x_p y_p}{2} & -\frac{c_2 y_p^2}{2} & -\frac{c_2 y_p^2}{2} \\
-\frac{c_1 x_p y_p - c_2 x_p y_p}{2} & -\frac{c_1 x_p y_p + c_2 x_p y_p}{2} & -\frac{c_1 x_p^2 + c_2 x_p^2}{2} & -\frac{c_2 y_p^2}{2} \\
-\frac{c_2 y_p^2}{2} & -\frac{c_2 y_p^2}{2} & 2EI / L & -\frac{c_2 y_p^2}{2} \\
\end{bmatrix}
\]
4-13 Constant-strain Triangular Element

One of the simplest two-dimensional elements to formulate in FEA is the constant-strain triangular (CST) or the linear triangular element.

The CST element is assumed to have three corner nodes \((i, j, k)\) with two translational DOF at each node (6 DOF for the element). The nodal displacement vector for the element is defined by:

\[
[u] = \begin{bmatrix} u_i & v_i & u_j & v_j & u_k & v_k \end{bmatrix}^T
\]
The locations of the nodes are defined by \( x \) and \( y \) coordinates relative to a global reference frame. The triangle can have arbitrary proportions as defined by the locations of its nodes.

### 4-14 Displacement Field:

To use the Rayleigh-Ritz method we need to assume displacement fields in the \( x \) and \( y \) directions such that we have exactly six undetermined coefficients. Complete displacement fields with six undetermined coefficients are:

\[
    u(x,y) = a_1 + a_2x + a_3y
\]
\[
    v(x,y) = a_4 + a_5x + a_6y
\]

**Note that** both functions vary with \( x \) and \( y \).

These relations can also be written as:

\[
    \begin{pmatrix}
        u(x,y) \\
        v(x,y)
    \end{pmatrix}
    =
    \begin{bmatrix}
        1 & x & y & 0 & 0 & 0 \\
        0 & 0 & 0 & 1 & x & y
    \end{bmatrix}
    \begin{pmatrix}
        a_1 \\
        a_2 \\
        a_3 \\
        a_4 \\
        a_5 \\
        a_6
    \end{pmatrix}
\]

More concisely,

\[
    \mathbf{u}(x) = \mathbf{X}(x)\mathbf{a}
\]

### 4-15 Shape Functions:

Since the assumed displacements must equal the nodal displacements at the three nodes we have:

\[
    \begin{bmatrix}
        1 & x_i & y_i & 0 & 0 & 0 \\
        0 & 0 & 0 & 1 & x_i & y_i \\
        1 & x_j & y_j & 0 & 0 & 0 \\
        0 & 0 & 0 & 1 & x_j & y_j \\
        1 & x_k & y_k & 0 & 0 & 0 \\
        0 & 0 & 0 & 1 & x_k & y_k
    \end{bmatrix}
    \begin{pmatrix}
        a_1 \\
        a_2 \\
        a_3 \\
        a_4 \\
        a_5 \\
        a_6
    \end{pmatrix}
    =
    \begin{pmatrix}
        u_i \\
        v_i \\
        u_j \\
        v_j \\
        u_k \\
        v_k
    \end{pmatrix}
\]
More concisely (as with other element types) 
\[ \bar{X}a = u \]

This results in: 
\[ a = \bar{X}^{-1}u \]

Substituting in the displacement field expression we obtain:

\[ \bar{u}(x) = X(x)\bar{X}^{-1}u = N(x)u \]

Where \( N(x) = N(x, y) \) is the shape function matrix for the CST element.

The shape function matrix that results is given by:

\[
N(x) = \frac{1}{2A} \begin{bmatrix}
N_1(x, y) & 0 & N_2(x, y) & 0 & N_3(x, y) & 0 \\
0 & N_1(x, y) & 0 & N_2(x, y) & 0 & N_3(x, y)
\end{bmatrix}
\]

Where \( A = \text{det} (\bar{X}) \) is the area of the triangle and:

\[
N_1(x, y) = \alpha_1 + \beta_1x + \gamma_1y \quad N_2(x, y) = \alpha_2 + \beta_2x + \gamma_2y \quad N_3(x, y) = \alpha_3 + \beta_3x + \gamma_3y
\]

\[
\alpha_1 = x_2y_3 - x_1y_2 \quad \alpha_2 = x_3y_1 - x_1y_3 \quad \alpha_3 = x_1y_2 - x_2y_1
\]

\[
\beta_1 = y_2 - y_3 \quad \beta_2 = y_3 - y_1 \quad \beta_3 = y_1 - y_2
\]

\[
\gamma_1 = x_3 - x_2 \quad \gamma_2 = x_1 - x_3 \quad \gamma_3 = x_2 - x_1
\]

**4-16 Strain Vector:**

The relationship obtained thus far can be written in concise form as:

\[ \bar{u}(x) = N(x)u \]
Recall that strains in a two-dimensional domain are given by:

\[
\begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{bmatrix} = \begin{bmatrix}
\frac{\partial}{\partial x} & 0 \\
0 & \frac{\partial}{\partial y} \\
\frac{\partial}{\partial y} & \frac{\partial}{\partial x}
\end{bmatrix} \begin{bmatrix}
u(x,y)
\end{bmatrix} \quad \text{Or} \quad \varepsilon = \partial \mathbf{u}(x)
\]

4-17 Strain Displacement Matrix:

Applying this definition to displacement field for the CST element we obtain:

\[
\varepsilon = \partial \mathbf{u}(x) = \partial \mathbf{N}(x) \mathbf{u} = \mathbf{B}(x) \mathbf{u}
\]

Where:

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

Performing the required differentiations \( \mathbf{B} \) is obtained more explicitly as:

\[
[B] = \frac{1}{2A} \begin{bmatrix}
\beta_1 & 0 & \beta_2 & 0 & \beta_3 & 0 \\
0 & \gamma_1 & 0 & \gamma_2 & 0 & \gamma_3 \\
\gamma_1 & \beta_1 & \gamma_2 & \beta_2 & \gamma_3 & \beta_3
\end{bmatrix}
\]
Where $\beta_i$ and $\gamma_i$ have the same definitions as those in $N$ Clearly the strain-displacement matrix $B$ is a constant matrix (no dependence on $x$ or $y$)

**4-18 Strain Energy:**

The strain energy of the CST element can now be formulated:

$$\Omega = \frac{1}{2} \int_V \epsilon^T \epsilon \, dV = \frac{1}{2} \left( \int_V B^T E B \, dV \right) u = \frac{1}{2} u^T k u$$

If the element has thickness $t$ at any point across its area $dV = tdA$. Since $B$ and $E$ are constant matrices and assuming $t$ is constant throughout the element

$$\int_A B^T E B \, dV = \int_A B^T E B \, t \, dA = B^T E B t \int_A dA = B^T E B t A$$

**4-19 Plane Stress and Plane Strain:**

Recall that for a state of plane stress

$$[E] = \frac{E}{1 - \nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1 - \nu}{2} \end{bmatrix}$$

And for a state of plane strain

$$[E] = \frac{E}{(1 + \nu)(1 - 2\nu)} \begin{bmatrix} 1 - \nu & \nu & 0 \\ \nu & 1 - \nu & 0 \\ 0 & 0 & \frac{1 - 2\nu}{2} \end{bmatrix}$$
4-19-1 Linear Strain Triangular Element:

A higher order but still simple two-dimensional element is the 6-node linear strain triangular (LST) or the quadratic triangular element.

4-20 LST Displacement Field:

Complete displacement fields for the LST triangles are;

\[ u(x, y) = a_1 + a_2 x + a_3 y + a_4 x^2 + a_5 xy + a_6 y^2 \]
\[ v(x, y) = a_7 + a_8 x + a_9 y + a_{10} x^2 + a_{11} xy + a_{12} y^2 \]

This field results in a shape function matrix \( N \) that is quadratic in \( x \) and \( y \). The quadratic shape function results in a strain-displacement matrix \( B \) that varies linearly with \( x \) and \( y \).

4-21 LST Stiffness Matrix:

Applying the same procedure as before for a constant thickness element we obtain a \( 12 \times 12 \) stiffness matrix given by;

\[
\mathbf{k} = t \int_{x_{\min}}^{x_{\max}} \int_{y_{\min}}^{y_{\max}} \mathbf{B}^T \mathbf{E} \mathbf{B} \, dx \, dy
\]
The integration shown is in general laborious to perform analytically; as a result a numerical method such as Gaussian quadrature is used to obtain the matrix

**4-22 Area Coordinates:**

Area or natural coordinates are defined in terms of area ratios.

\[ L_1 = \frac{A_1}{A} \]
\[ L_2 = \frac{A_2}{A} \]
\[ L_3 = \frac{A_3}{A} \]
\[ A = A_1 + A_2 + A_3 \]

**Quadratic in \( L_1, L_2, L_3 \)**

The area coordinates defined for a general triangle vary between 0 and 1 as the point \( P \) moves from an edge to an arbitrary point in the interior of the triangle. The shape functions can be expressed in terms of only \( L_1 \) and \( L_2 \) since the three coordinates are not independent.

\[ u = u_i (2L_1 - 1) L_1 + u_j (2L_2 - 1) L_2 + u_k (2L_3 - 1) L_3 + u_l (4L_1 L_2) + u_m (4L_2 L_3) + u_n (4L_3 L_1) \]

This transforms the expression for the stiffness matrix to

\[ \mathbf{K} = t \int_{0}^{1} \int_{0}^{1} \mathbf{B}^T \mathbf{E} \mathbf{B} dL_1 dL_2 \]

**4-23 Plane Quadrilaterals Bilinear Element:**
The 4-node quadrilateral element is the simplest four-sided two-dimensional element

**4-23-1 Bilinear Displacement Field:**

The assumed displacement field for this element is

\[
\begin{align*}
    u(x, y) &= a_1 + a_2 x + a_3 y + a_4 x y \\
    v(x, y) &= a_5 + a_6 x + a_7 y + a_8 x y
\end{align*}
\]

Or

\[
\begin{align*}
    \{u(x, y)\} &= \begin{bmatrix} 1 & x & y & xy & 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \\ a_5 \\ a_6 \\ a_7 \\ a_8 \end{bmatrix} \\
    \{v(x, y)\} &= \begin{bmatrix} 0 & 0 & 0 & 0 & 1 & x & y & xy \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \\ a_5 \\ a_6 \\ a_7 \\ a_8 \end{bmatrix}
\end{align*}
\]

Note that the assumed displacement field is not “complete” (neither linear nor quadratic). Writing these expressions more concisely and performing the usual operations we obtain;

\[
\bar{X}a = u \quad a = \bar{X}^{-1}u
\]

\[
\bar{u}(x) = X(x)\bar{X}^{-1}u = N(x)u
\]
Where \( \mathbf{N}(x) = \mathbf{N}(x, y) \) is the shape function matrix for the plane quadrilateral bilinear (PQB) element

\[
\mathbf{N}(x) = \begin{bmatrix}
N_1(x,y) & 0 & N_1(x,y) & 0 & N_3(x,y) & 0 & N_3(x,y) & 0 \\
0 & N_1(x,y) & 0 & N_1(x,y) & 0 & N_3(x,y) & 0 & N_3(x,y)
\end{bmatrix}
\]

### 4-23-2 PQB Strain-Displacement Matrix:

The strain in this element can now be computed from;

\[
\mathbf{\varepsilon} = \partial \mathbf{u}(x) = \partial \mathbf{N}(x) \mathbf{u} = \mathbf{B}(x) \mathbf{u}
\]

Where:

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

Note that because of the assumed displacement field;

\[
\varepsilon_x = \varepsilon_x(y) \\
\varepsilon_y = \varepsilon_y(x) \\
\gamma_{xy} = \gamma_{xy}(x, y)
\]
This strain distribution may be unsuitable for some applications. Using this matrix the stiffness matrix for a constant thickness $h$ PQB element can be computed.

$$\mathbf{k} = h \int_{x_{\min}}^{x_{\max}} \int_{y_{\min}}^{y_{\max}} \mathbf{B}^T \mathbf{E} \mathbf{B} \, dx \, dy$$

### 4-24 Natural Coordinates:

Natural coordinates for quadrilaterals are defined as;

This transforms the expression for the stiffness matrix to

$$\mathbf{k} = h \int_{-1}^{1} \int_{-1}^{1} \mathbf{B}^T \mathbf{E} \mathbf{B} \, ds \, dt$$

Additional mathematical operations required to transform strains and stresses from natural to global coordinates.

### 4-25 The static analysis program:
START

READ N,M

READ X(I), Y(I), EX(I), EY(I) & ER(I)
I = 1,N

IF I = N

WRITE (NF)

READ NUMBER OF LOADED NODES (LJ)

READ LOADS COMPOUND AT EACH OF THE LOADED NODES

READ PROBLEM TYPE CODES CC, PR

IF PR = 0

READ E, V

IF CO = 1

CALCULATE D1, D2, D3 (PLANE STRESS)

CALCULATE D1, D2, D3 (PLANE STRAIN)

ASSEMBLE NODEL FORCES & LOAD VECTOR

READ NO. OF GAUSS LEGENDRE POINTS (NCP)

1
4-26 Input and Output Data:

4-26-1 Input data for finite element problems:

The input data for the numerical analysis as follow:
1- Number of nodes.
   Number of elements.
2- X-Coordinates, Y-Coordinates, X-Restraint, Y-Restraint and R-Restraint for each node from the mesh. Using (0) degree of freedom is restrained and (1) degree of freedom is free.
3- Number of loaded nodes.
4- For each node (repeated as many as loaded nodes number) node number, X-Component of force, Y-Component of force and M-component.
5- Problem codes
   \[ \text{PR} \]
   \[ \begin{align*}
   \text{PR} &= 0 \text{ Constant } E, V \text{ for all elements.} \\
   \text{PR} &= 1 \text{ Variable } E, V
   \end{align*} \]
   \[ \text{CO} \]
   \[ \begin{align*}
   \text{CO} &= 1 \text{ (Plane stress problem).} \\
   \text{CO} &= 0 \text{ (Plane strain problem).}
   \end{align*} \]
6- Number of Gauss-Legendre quadrature points (NGP).
7- For each node:
   - Point coordinates.
   - Point weight.
8- Element type code (ET).
   \[ \text{ET} = 1 \text{ for beam element.} \]
   \[ \text{ET} = 6 \text{ for fracture element.} \]
   \[ \text{ET} = 5 \text{ for 4-node quad. Element.} \]
9- Body forces intensity
   \[ \text{BH- Horizontal} \]
   \[ \text{BV- Vertical} \]

If \[ \text{ET=1} \], it is required to input:
10-Connecting nodes.
11-Inertia, Area, E.
If ET=6, it is required to input:
12- Connecting nodes.
13- Inertia, Area, E, V, crack-depth, depth of lining, breadth of lining and crack code (TA = 1 crack upper and TA = 0 crack lower).
If ET=5, it is required to input:
14- E, V, TH.
15- Connecting nodes in anticlockwise direction.

4-26-2 Output data for finite element problems:

The output data for each run were:
1- Number of nodes and number of elements.
2- CO and PR.
3- E, V and TH for soil medium.
4- Half band width.
5- Nodal displacements in tabulated form (δx, δy and R).
6- End forces of beam element (N.F, S.F and B.M) if any.
7- End forces of fracture of beam element (N.F, S.F and B.M) if any.
8- Stress and strain for 4-node quad. Elements (σx, σy, τxy, εx, εy, γxy) in a tabulated form.

Example(1): Cantilever beam model by using finite element method (2D element).
A cantilevered beam 1 m long, 12 cm wide and 50 cm high is loaded by an end load of 10 ton. The Young’s modulus for the material is
2.1*10^6 kg/cm^2 model a section of the beam and compares the maximum stresses produced with the exact solution.

The maximum stress in the beam using “exact” methods is;

\[ \sigma_{x,\text{max}} = \frac{M \cdot Y}{I} = \frac{(10)(1)(10)(1*10^5)}{\frac{1}{12}(12)(50)^3} = 80 \text{ kg/cm}^2 \]

**Where:**
M = Maximum moment on the cantilever at the fixation.
Y = Height of the section / 2.
I = Second moment of inertia.

The maximum deflection (at the free end) using “exact” methods is;

\[ U_{\text{max.}} = \frac{FL^3}{3EI} = \frac{(10)(1)^3}{3(2.1*10^6)\frac{1}{12}(0.12)(0.50)^3} = 3.17*10^{-7} \text{ m} \]

When the beam is modeled as a set of plane stress elements the load at its end must be divided by its width to produce a load per unit width; thus the load
Applied at the end must be \((10/0.1 = 100 \text{ ton/m})\). Using mapped meshing (with an element side of 0.05 m) we obtain in ANSYS.

ANSYS produces the following solution for this model (contour plot of SX on deformed shape).

Using mapped meshing (with an element side of 0.05 m) we obtain in ANSYS.

ANSYS produces the following solution for this model (contour plot of SX on deformed shape).
The second mapped mesh element provides a better approximation with fewer elements and D.O.F than the first element in this case because the stress variation in the y direction is linear in the exact solution.

The first mapped mesh element may provide a better approximation in instances when the bending moment varies quadratically or at a higher order in the x direction.

**Example (2): Multistory frame with crack.**

![Diagram of a multistory frame with cracks](a)
(b) Test model dimension and crack location
(a) Tie above foundation level       (b) Tie in foundation level

The finite element mesh
Example(3): For the given multistory frame, construct the finite element mesh and write the input data file.
$E_s = 1400 \text{ t/m}^2$  
$E_c = 2 \times 10^6 \text{ t/m}^3$

Assume Frame Depth 60cm  
Foundation Depth 70cm

**Suitable Elements**
- Beam Element  198
- Friction Element  12
- 4NIQE  252
- Infinite Element  50
  2Node  48
  1Node  2

NN  462
NE  512
<p>| | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>462,450</td>
<td>9.0,3.0,1,1.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.0,0.0,0,0,0</td>
<td>10.0,3.0,1,1.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.0,0.0,1,0,0</td>
<td>11.0,3.0,1,1.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.0,0.0,1,0,0</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6.0, 0.0,1,0,0</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7.0, 0.0,1,0,0</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8.0,0.0,1,0,0</td>
<td>15.0,3.0,1,1.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9.0, 0.0,1,0,0</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10.0, 0.0,1,0,0</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11.0,0.0,1,0,0</td>
<td>24.0,36.0,1,1.1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12.0,0.0,1,0,0</td>
<td>25.0,36.0,1,1.1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>26.0,36.0,1,1.1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.0,3.0,0,1,0</td>
<td>30.0,36.0,1,1.1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.0,3.0,1,1,0</td>
<td>68</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.0,3.0,1,1,0</td>
<td>330,0.0,-1.0,0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6.0,3.0,1,1,0</td>
<td>331,0.0,-2.0,0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7.0,3.0,1,1,0</td>
<td>332,0.0,-2.0,0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8.0,3.0,1,1,0</td>
<td>333,0.0,-2.0,0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>334,0.0,-2.0,0.0</td>
<td>0.0,0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>338,0.0,-2.0,0.0</td>
<td>1,2,33,32</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>339,0.0,-2.0,0.0</td>
<td>5</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>340,0.0,-2.0,0.0</td>
<td>0.0,0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>341,0.0,-2.0,0.0</td>
<td>2,3,34,33</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>342,0.0,-1.0,0.0</td>
<td>5</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>343,0.0,-1.0,0.0</td>
<td>0.0,0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>3,4,35,34</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
348,0.0,-2.0,0.0
349,0.0,-2.0,0.0
350, 0.0,-2.0, 0.0
351, 0.0,-2.0, 0.0
462,0.0,-4.0,0.0
0, 1
1400, 0.333333333,1
5
0.0000000,0.5688888
0.5384093,0.4786286
-.5384093,0.4786286
0.9061798,0.2369268
-.9061798,0.2369268
5
0.0,0.0
194,195
0.0285,0.7,2000000
1
0.0,0.0
200,201

**Example (5):** For the given cantilever retaining wall, construct the finite element mesh and write the input data file.

\[ E_s = 3000 \text{ t/m}^2 \]
\[ E_s = 2800 \text{ t/m}^2 \text{ Back Fill} \]
\[ E_c = 2 \times 10^6 \text{ t/m}^2 \]
**Suitable Elements:**
- Beam Element 11
- Friction Element 5 " at the base of the R.W "
- 4NIQE 212
- Infinite Element 41
  - 2Node 39
  - 1Node 2

**Nodes:**
- NN 246
- NE 269

```
<table>
<thead>
<tr>
<th>1</th>
<th>1</th>
<th>1</th>
<th>1</th>
<th>1</th>
<th>1</th>
<th>1</th>
<th>1</th>
<th>1</th>
<th>1</th>
</tr>
</thead>
</table>
```

---

```
<table>
<thead>
<tr>
<th>246</th>
</tr>
</thead>
</table>
```

---

```
| 0.8t | 2t  |
| 6.0t |
| 1.3274t |
| 2.0476t |
| 1.0t |
| 3.0t |
| 0.125t |
| 40 |
| 20 |
| 60 |
| 40 |
| 20 |
| 2 | 2 | 2 |

---

```
| 21 |
| 22 |
| 23 |
| 24 |
| 25 |
| 26 |
| 2 |
| 3 |
| 4 |
| 5 |
| 6 |
| 7 |
| 8 |
| 9 |
| 10 |
| 1 |
| 1 |
| 1 |
| 1 |
| 1 |
| 1 |
| 1 |
```

---

```
| 0.8t | 2t  |
| 6.0t |
| 1.3274t |
| 2.0476t |
| 1.0t |
| 3.0t |
| 0.125t |
| 40 |
| 20 |
| 60 |
| 40 |
| 20 |
| 2 | 2 | 2 |

---

```
| 21 |
| 22 |
| 23 |
| 24 |
| 25 |
| 26 |
| 2 |
| 3 |
| 4 |
| 5 |
| 6 |
| 7 |
| 8 |
| 9 |
| 10 |
| 1 |
| 1 |
| 1 |
| 1 |
| 1 |
| 1 |
| 1 |
```
| 246,223 | . |
| 0.0,0.0,0,0,0 | 18.0,18.0,1,1,0 |
| 2.0,0.0,1,0,0 | 20.0,18.0,1,1,0 |
| 4.0,0.0,1,0,0 | 22.0,18.0,1,1,0 |
| . | 24.0,18.0,0,1,0 |
| . | 23 |
| 10.0,0.0,1,0,0 | 161.0,0.0,-3.0,0.0 |
| 11.0,0.0,1,0,0 | 162.0,0.0,-6.0,0.0 |
| 11.5,0.0,1,0,0 | 163.0,0.0,-6.0,0.0 |
| 0.0,3.0,0,1,0 | 164.0.0,-4.5,0.0 |
| 2.0,3.0,1,1,0 | 168.0.0,-3.0,0.0 |
| 4.0,3.0,1,1,0 | 169.0.0,-3.0,0.0 |
| . | 170.1.327,0.0,0.0 |
| 9.0,3.0,1,1,0 | 244.0.0,-4.0,0.0 |
| 10.0,3.0,1,1,0 | 245.0.0,-4.0,0.0 |
| 11.0,3.0,1,1,0 | 246.0.0,-4.0,0.0 |
| 11.5, 3.0,1,1,0 | . |
| 12.0, 3.0,1,1,0 | . |
| . | 1,0 |
| . | 5 |
| 10.0,12.0,1,1,1 | 0.0000000,0.5688888 |
| 11.0,12.0,1,1,1 | 0.5384093,0.4786286 |
| 11.5,12.0,1,1,1 | -0.5384093,0.4786286 |
| 12.0,12.0,1,1,1 | 0.9061798,0.2369268 |
| 5 |
| 0.0,0.0 | -0.9061798,0.2369268 |
| 3000,0.3333333333,1 | 459,460 |
| 1,2,22,21 | 0.0,0.0 |
| 5 |
| 0.0,0.0 | 460,461 |
| 3000,0.3333333333,1 |
| 2,3,23,22 |
| 5 |
| 0.0,0.0 |
| 3000,0.3333333333,1 |
| 5,6,26,25 |
| 1 |
| 0.0,0.0 |
| 168,169 |
| 0.042,0.80,2000000 |
| 1 |
| 0.0,0.0 |
| 169,170 |
| 0.042,0.80,2000000 |
| . |
| 5 |
| 0.0,0.0 |
| 169,170 |
| 0.042,0.80,2000000 |