A Project

Presented to the faculty of the Department of Civil Engineering

California State University, Sacramento

Submitted in partial satisfaction of the requirements for the degree of

MASTER OF SCIENCE

in

Civil Engineering

by

S M Imrul Kabir

FALL 2013
GENERAL FINITE ELEMENT CODE THEORY MANUAL

A Project

by

S M Imrul Kabir

Approved by:

__________________________________, Committee Chair
Dr. Matthew Salveson, P.E.

____________________________________
Date

ii
Student: S M Imrul Kabir

I certify that this student has met the requirements for format contained in the University format manual, and that this project is suitable for shelving in the Library and credit is to be awarded for the project.

____________________________, Department Chair
Dr. Kevan Shafizadeh, P.E., P.T.O.E

Department of Civil Engineering
Abstract

of

GENERAL FINITE ELEMENT CODE THEORY MANUAL

by

S M Imrul Kabir

The finite element method (FEM) is a numerical technique for solving problems which are described by partial differential equations or can be formulated as functional minimization. A domain of interest is represented as an assembly of finite elements. Approximating functions in finite elements are determined in terms of nodal values of a physical field which is sought. A continuous physical problem is transformed into a discretized finite element problem with unknown nodal values. The key equation for solving finite element problems is \( \{\text{Force}\} = [\text{Stiffness}] \{\text{Displacement}\} \). Dimension and values of force vector, stiffness matrix and displacement vector varies for different element types. Due to its large computational size finite element problem needs a computer program to be solved.

*General Finite Element Code* (GFEC) is a type of a computer program that uses the finite element method to analyze a material or an object and find how applied stresses will affect the material or the design. In order to illustrate computer implementation of FEM, General Finite Element Code (GFEC) program has been developed in FORTRAN language. Different elements have been incorporated in this computer program. Out of those elements, following elements have been discussed in this manual.

a) 3-node plane stress element
b) 4-node plane stress element

c) 4-node tetrahedral element

d) Nearly incompressible 2D plane stress element

e) Lumped plasticity frame element

Theory and solution process for these elements have been collected from various books and journals. Collected information have been included and organized in this manual in such a way so that reading this theory manual, users of GFEC can understand the *behind the scenario process*.
ACKNOWLEDGEMENTS

I would like to express my deepest gratitude to all those who provided me the possibility to complete this report. A special appreciation I give my project supervisor, Dr. Matthew Salveson, P.E., whose involvement in stimulating suggestions and supports, helped me to complete my project especially in writing this report.

In addition, I would like to thank my parents, siblings and friends who helped me a lot in finalizing this project providing me necessary supports during this work period. Also, thanks to Mr. Ratan Huda, a structural engineer at NY DOT, who has assisted me in proof reading works.
TABLE OF CONTENTS

Acknowledgements .................................................................................................................. vi
List of Figures ....................................................................................................................... viii

Chapter

1. FINITE ELEMENT ANALYSIS ..................................................................................... 01
2. THREE NODE PLANE STRESS ELEMENTS ............................................................... 03
3. FOUR NODE PLANE STRESS ELEMENTS ................................................................. 16
4. FOUR NODE TETRAHEDRON ELEMENTS ............................................................... 27
5. FINITE ELEMENT ANALYSIS OF NEARLY INCOMPRESSIBLE MATERIAL ...... 35
6. LUMPED PLASTICITY MODEL OF FRAME ELEMENT........................................... 42

Work Cited ............................................................................................................................. 47
## LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figures</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.01</td>
<td>Basic triangular element showing degrees of freedom</td>
<td>4</td>
</tr>
<tr>
<td>2.02</td>
<td>Typical Flow Chart for element tangent calculation for 3 node element</td>
<td>14</td>
</tr>
<tr>
<td>2.03</td>
<td>Typical Flow chart for stress calculation for 3 node element</td>
<td>15</td>
</tr>
<tr>
<td>3.01</td>
<td>Basic four-node rectangular element with nodal degrees of freedom</td>
<td>16</td>
</tr>
<tr>
<td>3.02 (a)</td>
<td>Linear square element in s-t coordinates</td>
<td>20</td>
</tr>
<tr>
<td>3.02 (b)</td>
<td>Square element mapped into quadrilateral in x-y coordinates</td>
<td>20</td>
</tr>
<tr>
<td>3.03</td>
<td>Element Tangent for 4 Node Element programming steps</td>
<td>25</td>
</tr>
<tr>
<td>3.04</td>
<td>Element Stress for 4 Node Element programming steps</td>
<td>26</td>
</tr>
<tr>
<td>4.01</td>
<td>Tetrahedron element</td>
<td>27</td>
</tr>
<tr>
<td>4.02</td>
<td>Four node tetrahedral solid element</td>
<td>28</td>
</tr>
<tr>
<td>5.01</td>
<td>Material’s Stiffness vs. its Poisson ratio</td>
<td>36</td>
</tr>
<tr>
<td>6.01</td>
<td>Flat surfaces approximating the elastic domain for the end sections</td>
<td>43</td>
</tr>
</tbody>
</table>
CHAPTER 01

FINITE ELEMENT ANALYSIS

The finite element method (FEM) is a numerical technique for solving boundary value problems defined by partial differential equations. Structural engineers use finite element analysis (FEA), the practical application of the finite element method (FEM), to model structural problems mathematically and solve those numerically. For analysis purpose, a domain of interest is divided into a number of pieces namely finite elements. Then equilibrium equations for all elements are formed in terms of known and unknown nodal values. Generally, these equations are solved by matrix operations. Computer programming is must for this kind of mathematical operations.

Shape of an element is defined by a system of points, called ‘nodes’. All elements within a domain are connected with each other through these nodes. In this way, the domain is modeled as a mesh of finite elements. Nodes in the mesh contain the material and structural properties of the model. These properties define how the model will react to certain conditions. The model can be created using one-dimensional, two-dimensional or three-dimensional elements.

In the book ‘Introduction to Finite Element’, G. P. Nishikov has listed out main steps of the finite element solution procedure described below:

1. Discretize the continuum. The first step is to divide a solution region into finite elements. Usually a preprocessor program is used for finite element meshing. The description of the mesh consists of several arrays main of which are nodal coordinates and element connectivities.

2. Select interpolation functions. Interpolation functions are used to interpolate the field variables over the element. Often, polynomials are selected as interpolation functions. The degree of the polynomial depends on the number of nodes assigned to the element.
3. *Find the element properties.* The matrix equation for the finite element should be established which relates the nodal values of the unknown function to other parameters. For this task, different approaches can be used; the most convenient are: the variational approach and the Galerkin method.

4. *Assemble the element equations.* To find the global equation system for the whole solution region we must assemble all the element equations. In other words, we must combine local element equations for all elements used for discretization. Element connectivities are used for the assembly process. Before the solution, boundary conditions (which are not accounted in element equations) should be imposed.

5. *Solve the global equation system.* The finite element global equation system is typically sparse, symmetric and positive definite. Direct and iterative methods can be used for solution. The nodal values of the sought function are produced as a result of the solution.

6. *Compute additional results.* In many cases, we need to calculate additional parameters. For example, in mechanical problems strains and stresses are of interest in addition to displacements, which are obtained after solution of the global equation system.

(Nikishkov, 2007).
CHAPTER 02
THREE NODE PLANE STRESS ELEMENTS

It is required to subdivide an object into many small pieces to carry out Finite Element Analysis of that object. Pieces are referred as elements. Elements can be triangle or rectangle elements. A good approximation can be obtained for the objects having irregular shaped boundary using triangular elements. To determine nodal values in a triangular element D. L. Logan has presented, in his book *A First Course in the Finite Element Method*, a calculation process described as follows:

To demonstrate the calculation process let us consider a basic triangular element as shown in figure 2.01. The element has three nodes $i(x_i, y_i)$, $j(x_j, y_j)$ and $m(x_m, y_m)$. Each node has two degrees of freedom. $u_i$ and $v_i$ represent the node $i$ displacement components in the $x$ and $y$ directions, respectively. Similarly, $u_j$ and $v_j$ represent the node $j$ displacement components in the $x$ and $y$ directions, respectively. In the same way, $u_m$ and $v_m$ represent the node $m$ displacement components in the $x$ and $y$ directions, respectively. Now let us consider a load $T_s$ is being applied in the horizontal direction on the plate from which the element has been taken. This load will pass through the body and hence will create displacement in every single point of the plate. Unknown forces and deflections in each node of meshed element are calculated from known forces and displacements. Then mapping technique is used to determine forces and deflections in the whole body.
The nodal displacement can be expressed as a column matrix as below:

\[
\{d\} = \begin{bmatrix} d_i \\ d_j \\ d_m \end{bmatrix} = \begin{bmatrix} u_i \\ v_i \\ u_j \\ v_j \\ u_m \\ v_m \end{bmatrix}
\]  

(2.01)

At any interior point \((x,y)\) of the element linear displacement functions are as below:

\[
u(x, y) = a_1 + a_2 x + a_3 y \\
u(x, y) = a_4 + a_5 x + a_6 y
\]

(2.02)

We can rearrange the equation (2.02) and the general displacement as a product of two matrices.

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

(2.03)
Where \( \{u_i\} \) denotes the general displacement. Replacing the coordinate values of the nodes into equation (2.02) we obtain

\[
\begin{align*}
  u_i &= u(x_i, y_i) = a_1 + a_2 x_i + a_3 y_i \\
  u_j &= u(x_j, y_j) = a_1 + a_2 x_j + a_3 y_j \\
  u_m &= u(x_m, y_m) = a_1 + a_2 x_m + a_3 y_m \\
  v_i &= v(x_i, y_i) = a_4 + a_5 x_i + a_6 y_i \\
  v_j &= v(x_j, y_j) = a_4 + a_5 x_j + a_6 y_j \\
  v_m &= v(x_m, y_m) = a_4 + a_5 x_m + a_6 y_m
\end{align*}
\]  

(2.04)

From first three of equations (2.04) we can express the displacement matrix \( \{u\} \) as

\[
\begin{pmatrix}
  u_i \\
  u_j \\
  u_m
\end{pmatrix} =
\begin{bmatrix}
  1 & x_i & y_i \\
  1 & x_j & y_j \\
  1 & x_m & y_m
\end{bmatrix} \begin{pmatrix}
  a_1 \\
  a_2 \\
  a_3
\end{pmatrix}
\]

(2.05)

So we can now obtain the values for \( a \)'s solving equation (2.05),

\[
\{a\} = [x]^{-1}\{u\}
\]

(2.06)

Using method of cofactors the inverse of \([x]\) can be calculated as shown in below,

\[
[x]^{-1} = \frac{1}{2A} \begin{bmatrix}
  \alpha_i & \alpha_j & \alpha_m \\
  \beta_i & \beta_j & \beta_m \\
  \gamma_i & \gamma_j & \gamma_m
\end{bmatrix}
\]

(2.07)

where

\[
2A = \begin{vmatrix}
  1 & x_i & y_i \\
  1 & x_j & y_j \\
  1 & x_m & y_m
\end{vmatrix}
\]

(2.08)

or,

\[
2A = x_i(y_j - y_m) + x_j(y_m - y_i) + x_m(y_i - y_j)
\]

(2.09)

Here \( A \) is the area of the triangle, and
We can now calculate $a_1$, $a_2$, $a_3$ solving the matrix equation,

\[
\begin{bmatrix}
    a_1 \\
    a_2 \\
    a_3
\end{bmatrix} = \frac{1}{2A} \begin{bmatrix}
    \alpha_i & \alpha_j & \alpha_m \\
    \beta_i & \beta_j & \beta_m \\
    \gamma_i & \gamma_j & \gamma_m
\end{bmatrix} \begin{bmatrix}
    u_i \\
    u_j \\
    u_m
\end{bmatrix}
\]

(2.11)

Similarly, we can obtain the values of $a_4$, $a_5$, $a_6$ using the last three of equation (2.04)

\[
\begin{bmatrix}
    a_4 \\
    a_5 \\
    a_6
\end{bmatrix} = \frac{1}{2A} \begin{bmatrix}
    \alpha_i & \alpha_j & \alpha_m \\
    \beta_i & \beta_j & \beta_m \\
    \gamma_i & \gamma_j & \gamma_m
\end{bmatrix} \begin{bmatrix}
    v_i \\
    v_j \\
    v_m
\end{bmatrix}
\]

(2.12)

If the first of equations (2.02) expressed in matrix form, we have

\[
\{a\} = [1 \times y] \begin{bmatrix}
    a_1 \\
    a_2 \\
    a_3
\end{bmatrix}
\]

(2.13)

Replacing the value for $a$’s from equation (2.11) into equation (2.13), we obtain

\[
\{u\} = \frac{1}{2A} [1 \times y] \begin{bmatrix}
    \alpha_i & \alpha_j & \alpha_m \\
    \beta_i & \beta_j & \beta_m \\
    \gamma_i & \gamma_j & \gamma_m
\end{bmatrix} \begin{bmatrix}
    u_i \\
    u_j \\
    u_m
\end{bmatrix}
\]

(2.14)

After multiplying right most two matrices in above equation we have

\[
\{u\} = \frac{1}{2A} [1 \times y] \begin{bmatrix}
    \alpha_i u_i + \alpha_j u_j + \alpha_m u_m \\
    \beta_i u_i + \beta_j u_j + \beta_m u_m \\
    \gamma_i u_i + \gamma_j u_j + \gamma_m u_m
\end{bmatrix}
\]

(2.15)

Multiplying the two matrices in equation (2.15) and rearranging, we obtain the $x$ displacement as below:
In the same way, we have the $y$ displacement given by

$$v(x, y) = \frac{1}{2A} \left\{ (\alpha_i + \beta_i x + \gamma_i y) v_i + (\alpha_j + \beta_j x + \gamma_j y) v_j + (\alpha_m + \beta_m x + \gamma_m y) v_m \right\}$$

(2.17)

We can rewrite equations (2.16) and (2.17) in a simpler form,

$$u(x, y) = N_i u_i + N_j u_j + N_m u_m$$

(2.18a)

$$v(x, y) = N_i v_i + N_j v_j + N_m v_m$$

(2.18b)

Where,

$$N_i = \frac{1}{2A} (\alpha_i + \beta_i x + \gamma_i y)$$

$$N_j = \frac{1}{2A} (\alpha_j + \beta_j x + \gamma_j y)$$

$$N_m = \frac{1}{2A} (\alpha_m + \beta_m x + \gamma_m y)$$

(2.19)

Expressing equations (2.18a) and (2.18b) in the matrix form, we have

$$\begin{bmatrix} \{u\} \\ \{v\} \end{bmatrix} = \begin{bmatrix} u(x, y) \\ v(x, y) \end{bmatrix} = \begin{bmatrix} N_i u_i + N_j u_j + N_m u_m \\ N_i v_i + N_j v_j + N_m v_m \end{bmatrix}$$

Or,

$$\{u\} = \begin{bmatrix} N_i & 0 & N_j & 0 & N_m & 0 \\ 0 & N_i & 0 & N_j & 0 & N_m \end{bmatrix} \begin{bmatrix} u_i \\ v_i \\ u_j \\ v_j \\ u_m \\ v_m \end{bmatrix}$$

(2.20)
Above matrix equation can be abbreviated as,

\[
\{\psi\} = [N]\{d\}
\]  

(2.21)

Where \([N]\) is given by

\[
[N] = \begin{bmatrix}
N_i & 0 & N_j & 0 & N_m & 0 \\
0 & N_i & 0 & N_j & 0 & N_m
\end{bmatrix}
\]  

(2.22)

\(N_i, N_j\) and \(N_m\) are the shape functions which represent the shape of the domain. At any point on the surface of the element

\[N_i + N_j + N_m = 1\]

At node \(i\) we must have \(N_i=1, N_j=0\) and \(N_m=0\). Similarly at node \(j\) we must have \(N_i=0, N_j=1\) and \(N_m=0\). Likewise at node \(m\) we must have \(N_i=0, N_j=0\) and \(N_m=1\). Displacement functions have been expressed in terms of shape functions. The element strains can be obtained the derivatives of unknown nodal displacements.

The strains are given by

\[
\{\varepsilon\} = \begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{bmatrix} = \begin{bmatrix}
\frac{\partial u}{\partial x} \\
\frac{\partial v}{\partial y} \\
\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}
\end{bmatrix}
\]  

(2.23)

Differentiating both side of the equation (2.18a) with respect to \(x\), we have
\[
\frac{\partial u}{\partial x} = u_x = \frac{\partial}{\partial x}(N_i u_i + N_j u_j + N_m u_m)
\]  \hspace{1cm} (2.24)

Since \( u_i = u(x_i, y_i) \) is a constant value, \( u_{i,x} = 0 \). In the same way \( u_{j,x} = 0, u_{m,x} = 0 \). So we can write the equation (2.24) as,

\[
u_{x} = N_{i,x} u_i + N_{j,x} u_j + N_{m,x} u_m
\]

(2.25)

The derivatives of the shape functions can be obtained from equation (2.19) as follows:

\[
N_{i,x} = \frac{1}{2A} \frac{\partial}{\partial x}(\alpha_i + \beta_i x + \gamma_i y) = \frac{\beta_i}{2A}
\]

(2.26)

\[
N_{j,x} = \frac{\beta_j}{2A} \quad \text{and} \quad N_{m,x} = \frac{\beta_m}{2A}
\]

(2.27)

Substituting values of equation (2.26) and (2.27) in equation (2.25), we have

\[
\frac{\partial u}{\partial x} = \frac{1}{2A} \left(\beta_i u_i + \beta_j u_j + \beta_m u_m\right)
\]

(2.28)

Likewise, we have

\[
\frac{\partial v}{\partial y} = \frac{1}{2A} \left(\gamma_i v_i + \gamma_j v_j + \gamma_m v_m\right)
\]

(2.29a)

\[
\frac{\partial u}{\partial y} + \frac{\partial v}{\partial y} = \frac{1}{2A} \left(\gamma_i u_i + \beta_i v_i + \gamma_j u_j + \beta_j v_j + \gamma_m u_m + \beta_m v_m\right)
\]

(2.29b)

Using equation (2.28), (2.29a) and (2.29b) in equation (2.23), we get
Equation (2.31) can be written in short form as

\[ \{ \varepsilon \} = [B] \{ d \} \]

(2.33)

Where

\[ [B] = [B_1 \ B_2 \ B_3] \]

(2.34)

For two dimensional elements stress/strain relationship is given by

\[ \begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} = [D] \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} \]

(2.35)
If $E$ is the modulus of elasticity and $\nu$ is poison’s ratio, then $[D]$ is given by

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

(2.36)

Substituting values from equation (2.33) in equation (2.35), we obtain

$$
\{\sigma\} = [D][B]\{d\}
$$

(2.37)

The stresses $\{\sigma\}$ are constant all over the element.

We can now derive the Element Stiffness Matrix and equations for a typical constant-strain triangular element using the principle of minimum potential energy.

Total potential energy is given by

$$
\pi_p = U + \Omega_b + \Omega_p + \Omega_z
$$

(2.38)

Where the strain energy $U$ is given by

$$
U = \frac{1}{2} \int_V \{\varepsilon\}^T [D]\{\varepsilon\}dV
$$

(2.39)

or, using equation (2.35), we have

$$
U = \frac{1}{2} \int_V \{\varepsilon\}^T [D]\{\varepsilon\}dV
$$

(2.40)

If $\{X\}$ is the body weight/unit volume, then potential energy of the body forces is
The potential energy of concentrated loads is given by

\[ \Omega_3 = -\{d\}^T \{P\} \]  

(2.42)

where \(\{d\}\) is the usual nodal displacements, and \(\{P\}\) is the external loads.

The potential energy of surface tractions is given by

\[ \Omega_2 = -\iint_S \{\psi_3\}^T \{T_s\} \, ds \]  

(2.43)

Using equations (2.38) - (2.43), we can produce

\[ \pi_p = \frac{1}{2} \iint \{d\}^T \{D\} \{B\} [d] \, dV - \iint \{d\}^T \{N\}^T \{X\} \, dV - \{d\}^T \{P\} - \iint \{d\}^T \{N_s\}^T \{T_s\} \, ds \]  

(2.44)

As the nodal displacements \(\{d\}\) are independent of the general \(x-y\) coordinates, equation (2.44) can be rewritten as

\[ \pi_p = \frac{1}{2} \{d\}^T \iint \{B\}^T \{D\} [B] [d] \, dV - \{d\}^T \iint \{N\}^T \{X\} \, dV - \{d\}^T \{P\} - \{d\}^T \iint \{N_s\}^T \{T_s\} \, ds \]  

(2.45)

where the second term, third term and fourth term of the right side of the equation (2.45) represent the body force, the concentrated nodal force and the surface traction respectively.

The total load system \(\{f\}\) on an element is,

\[ \{f\} = \iint \{N\}^T \{X\} \, dV + \{P\} + \iint \{N_s\}^T \{T_s\} \, ds \]  

(2.46)

Putting values from equation (2.46) in equation (2.45), we get
The partial derivative of $\pi_2$ with respect to the nodal displacements is

$$\frac{\partial \pi_2}{\partial \{d\}} = \left[ \int \int \int [B]^T \left[ D \int B \right] dV \{d\} - \{f\} \right] = 0$$

(2.48)

From equation (2.48), we can write

$$\left[ \int \int \int [B]^T \left[ D \int B \right] dV \{d\} = \{f\} \right]$$

(2.49)

From equation (2.49) we can see that

$$[k] = \int \int \int [B]^T \left[ D \int B \right] dV$$

(2.50)

For an element with constant thickness, $t$, equation (2.50) becomes

$$[k] = t \int \int [B]^T \left[ D \int B \right] dx dy$$

(2.51)

Performing the integration at the right side of the equation (2.51) becomes

$$[k] = tA[B]^T[D][B]$$

(2.52)

The nodal displacements can be obtained solving the system of algebraic equations given by

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

(2.53)

where, $\{d\}$ is the displacement matrix, and $[F]$ is the force matrix (Logan, 2007).

Typical programming algorithms for element tangent formation and stress calculation have been presented in figure (2.02) and figure (2.03) respectively.
Figure 2.02: Typical Flow Chart for element tangent calculation for 3 node element
**Figure 2.03**: Typical Flow chart for stress calculation for 3 node element

1. START
2. Read Inputs: node coordinates (x1,x2,x3,y1,y2,y3); Modulus of Elasticity (e_mod); Poison's ratio(nu), Displacements (utot)
3. Calculate area, coordinate of centroid
4. Call elm11_e_matrix
5. Call elm11_b_matrix
6. DB_matrix = e_matrix * b_matrix
7. q = DB_matrix * utot
8. Transfer values of q into gmsh_stress
9. FINISH
CHAPTER 03

FOUR NODE PLANE STRESS ELEMENTS

It is convenient to use triangular elements when the continuum has irregular shaped boundaries. On the other hand, using rectangular elements has two advantages – ease of data input and simpler interpretation of output stresses. Each node of a rectangular element has two degrees of freedom and hence an element has total eight degrees of freedom. Calculation steps to get the element stiffness matrix and associated equations are same as the steps for three node elements. In the book ‘A First Course in Finite Element Analysis’, D. L. Logan has presented the calculation process as follow:

Figure 3.01 shows a rectangular element. All of four nodes of this element have been marked with numerical numbers 1, 2, 3 and 4. A counterclockwise sequence has been maintained to avoid negative area count. Assume that base and height dimensions of the element are 2b and 2h, respectively. Nodal x displacements and y displacements are denoted with $u$ and $v$ respectively.

Figure 3.01: Basic four-node rectangular element with nodal degrees of freedom (Logan, 2007).
All eight nodal displacements are arranged in a matrix form as shown in equation (3.01).

\[
\{d\} = \begin{bmatrix}
u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 
\end{bmatrix}
\]  
(3.01)

At any point linear displacement functions can be selected as

\[
u(x, y) = a_1 + a_2 x + a_3 y + a_4 xy \\
v(x, y) = a_1 + a_2 x + a_3 y + a_4 xy
\]  
(3.02)

Eliminating the \(a_i\)'s from equation (3.02) and rearrange, we get

\[
u(x, y) = \frac{1}{4bh} \left[ (b - x)(h - y)u_1 + (b + x)(h - y)u_2 + (b + x)(h + y)u_3 + (b - x)(h + y)u_4 \right] \\
v(x, y) = \frac{1}{4bh} \left[ (b - x)(h - y)v_1 + (b + x)(h - y)v_2 + (b + x)(h + y)v_3 + (b - x)(h + y)v_4 \right]
\]  
(3.03)

Equation (3.03) can be rewritten in the matrix form as follow,

\[
\begin{bmatrix}u \\
v
\end{bmatrix} = \begin{bmatrix}N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\
0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4
\end{bmatrix}\begin{bmatrix}u_1 \\
v_1 \\
u_2 \\
v_2 \\
u_3 \\
v_3 \\
u_4 \\
v_4
\end{bmatrix}
\]  
(3.04)

where \([N]\) given by
Right side of the equation (3.03) is the product of the shape functions and unknown nodal displacements. Simplified form of this equation is

\[ \{ \psi \} = [N]\{d\} \]  

(3.04)

Where \( \{ \psi \} \) is the function of displacements. At node \( i \) the shape function, \( N_i = 1 \) and value of all other shape functions are zero. Same conditions needed for all other nodes. Now the element strains are calculated from the derivatives of displacement functions. For a plane stress the element strain function is given by

\[
\{ \varepsilon \} = \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{xy} \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \end{bmatrix}
\]

(3.05)

We can obtain derivatives of \( u \) and \( v \) using equation (3.04). Substituting the derivatives of \( u \) and \( v \) in equation (3.05), we get

\[ \{ \varepsilon \} = [B]\{d\} \]  

(3.06)

Where

\[
[B] = \frac{1}{4bh} \begin{bmatrix}
-(h-y) & 0 & (h-y) & 0 & (h+y) & 0 & -(h+y) & 0 \\
0 & -(b-x) & 0 & -(b+x) & 0 & (b+x) & 0 & (b-x) \\
-(b-x) & -(h-y) & -(b+x) & (h-y) & (b+x) & (h+y) & (b-x) & -(h+y)
\end{bmatrix}
\]  

(3.07)
The stresses are again given by equation (2.37) as shown in the previous chapter.

\[ \{\sigma\} = [D][B]\{d\} \]  

(2.37)

where \([B]\) is now that of equation (3.07) and \(\{d\}\) is that of equation (3.01).

The stiffness matrix is determined by

\[ [K] = \int_{-b}^{b} \int_{-h}^{h} [B]^T[D][B] \, dx \, dy \]  

(3.08)

where \([D]\) is again given by the equation (2.36) as shown in chapter 02. This is same for all plane stress or plane strain elements.

The element force matrix is given by

\[ \{f\} = \int \int [N]^T \{X\} \, dV + \int \int [N]^T \{P\} \, dS + \int \int [N]^T \{T\} \, dS \]  

(3.09)

where \([N]\) is a 2 x 8 matrix same as in equation (3.04). Then the element equation is,

\[ \{f\} = [K]\{d\} \]  

(3.10)

Now the isoparametric formulation for the Plane Element Stiffness Matrix is required.

Consider a quadratic plane element (figure 3.02 (a)) with eight degrees of freedom \(u_1, v_1, \ldots, u_4, v_4\) associated with the global \(x\) and \(y\) directions. Element’s geometry is defined by natural coordinate system \(s-t\). The corner nodes and the edges of quadrilateral are bounded by +1 or -1. Origin is at the center of the element.
Figure 3.02 (a) Linear square element in $s$-$t$ coordinates and (b) square element mapped into quadrilateral in $x$-$y$ coordinates whose size and shape are determined by the eight nodal coordinates $x_1, y_1, \ldots, y_4$ (Logan, 2007).

Let $(x_c, y_c)$ is the centroid of the element. Then the relationship between $s$-$t$ coordinates and the global element coordinates $x$ and $y$ is given by

$$x = x_c + bs \quad y = y_c + bt$$

(3.11)

Shape functions as shown in equation (3.05) is used to map the square of figure 3.02(a) in natural coordinate system to the quadrilateral in $x$ and $y$ coordinates as in figure 3.02(b). Size and shape of the quadrilateral are determined by the eight nodal coordinates $x_1, y_1, \ldots, x_4, y_4$. That is,

$$x = a_1 + a_2 s + a_3 t + a_4 st$$

$$y = a_5 + a_6 s + a_7 t + a_8 st$$

(3.12)

After solving for the $a_i$'s in terms of $x_1, x_2, x_3, x_4, y_1, y_2, y_3,$ and $y_4$, equation (3.12) becomes as,
We can rearrange equation (3.13) in matrix form, as below

\[
x = \frac{1}{4}[(1-s)(1-t)x_1 + (1+s)(1-t)x_2 + (1+s)(1+t)x_3 + (1-s)(1+t)x_4]
\]
\[
y = \frac{1}{4}[(1-s)(1-t)y_1 + (1+s)(1-t)y_2 + (1+s)(1+t)y_3 + (1-s)(1+t)y_4]
\]

We can rearrange equation (3.13) in matrix form, as below

\[
\begin{bmatrix} x \\ y \end{bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix} \begin{bmatrix} x_1 \\ y_1 \\ x_2 \\ y_2 \\ x_3 \\ y_3 \\ x_4 \\ y_4 \end{bmatrix}
\]

(3.14)

Shape functions in equation (3.14) are now calculated using \( s \) and \( t \),

\[
N_1 = \frac{(1-s)(1-t)}{4}
\]
\[
N_2 = \frac{(1+s)(1-t)}{4}
\]
\[
N_3 = \frac{(1+s)(1+t)}{4}
\]
\[
N_4 = \frac{(1-s)(1+t)}{4}
\]

(3.15)

At node 1 of the square element \( s = -1 \) and \( t = -1 \). Using these values for natural coordinates in equation (3.15) and (3.14), we have

\[
x = x_1 \quad y = y_1
\]

(3.16)

In the same way, we can map all nodes of the square element in \( s-t \) isoparametric coordinates into a quadrilateral element in global coordinates. Similarly the displacement functions are defined by the same shape functions as,
Strains of element are determined taking derivatives of the displacement functions. Displacement function is represented by \( f \) and it is a function of \( s \) and \( t \) as shown in equation (3.17). We can write,

\[
\frac{\partial f}{\partial s} = \frac{\partial f}{\partial x} \frac{\partial x}{\partial s} + \frac{\partial f}{\partial y} \frac{\partial y}{\partial s}
\]

\[
\frac{\partial f}{\partial t} = \frac{\partial f}{\partial x} \frac{\partial x}{\partial t} + \frac{\partial f}{\partial y} \frac{\partial y}{\partial t}
\]

(3.18)

Using Cramer’s rule equation (3.81) can be solved for \((\frac{\partial f}{\partial x})\) and \((\frac{\partial f}{\partial y})\), as

\[
\frac{\partial f}{\partial x} = \begin{vmatrix}
\frac{\partial f}{\partial s} & \frac{\partial f}{\partial t} \\
\frac{\partial x}{\partial s} & \frac{\partial x}{\partial t}
\end{vmatrix}
\]

\[
\frac{\partial f}{\partial y} = \begin{vmatrix}
\frac{\partial f}{\partial s} & \frac{\partial f}{\partial t} \\
\frac{\partial y}{\partial s} & \frac{\partial y}{\partial t}
\end{vmatrix}
\]

(3.19)

The determinant in the denominator is the determinant of Jacobian matrix \( J \).

We can express the element strains as

\[
\varepsilon = D' N d
\]

(3.20)

Where \( D' \) is an operator matrix given by
The product of shape function matrix and $D'$ matrix is defined as the $B$ matrix.

\[
B = D'N
\]

Substituting values from equation (3.21) and values from equation (3.15) into equation (3.22), we have

\[
B(s,t) = \frac{1}{|J|} \begin{bmatrix}
B_1 & B_2 & B_3 & B_4 
\end{bmatrix}
\]

where the sub-matrices $B_i$ ($i = 1, 2, 3, 4$) are given by

\[
B_i = \begin{bmatrix}
a(N_{i,t}) - b(N_{i,t}) & 0 \\
0 & c(N_{i,t}) - d(N_{i,t}) \\
c(N_{i,t}) - d(N_{i,t}) & a(N_{i,t}) - b(N_{i,t})
\end{bmatrix}
\]

and

\[
a = \frac{1}{4} \left[y_1(s-1) + y_2(-1-s) + y_3(1+s) + y_4(1-s)\right]
\]
\[
b = \frac{1}{4} \left[y_1(t-1) + y_2(1-t) + y_3(1+t) + y_4(-1-t)\right]
\]
\[
c = \frac{1}{4} \left[x_1(t-1) + x_2(1-t) + x_3(1+t) + x_4(-1-t)\right]
\]
\[
d = \frac{1}{4} \left[x_1(s-1) + x_2(-1-s) + x_3(1+s) + x_4(1-s)\right]
\]

Values for shape functions can be obtained from equation (3.15). We have

\[
N_{i,t} = \frac{1}{4}(t-1)
\]
The determinant \( |J| \) is a function of \( s \) and \( t \). It is evaluated as,

\[
|J| = \frac{1}{8} \{X_e\}^T \begin{bmatrix}
0 & 1-t & t-s & s-1 \\
 t-1 & 0 & s+1 & -s-t \\
 s-t & -s-1 & 0 & t+1 \\
 1-s & s+t & -t-1 & 0 \\
\end{bmatrix} \{Y_e\}
\]

where

\[
\{X_e\}^T = [x_1 \quad x_2 \quad x_3 \quad x_4]
\]

and

\[
\{Y_e\} = \begin{bmatrix}
y_1 \\
y_2 \\
y_3 \\
y_4 \\
\end{bmatrix}
\]

Now we can calculate the element stress using equation (2.37).

\[
\sigma = DBd
\]

Stress matrix is a function of \( s \) and \( t \) same as the \( B \) matrix (Logan, 2007).

Programming algorithms for calculating stiffness matrix and element stress have been presented in figure (3.03) and figure (3.04) respectively.
Figure 3.03: Element Tangent for 4 Node Element programming steps
Figure 3.04: Element Stress for 4 Node Element programming steps
This chapter deals with the basic formulation of three-dimensional elements. The simplest three-dimensional continuum is a tetrahedron, a polyhedron composed of four triangular faces, three of which meet at each corner or node. It has six edges and four nodes. Figure 4.01 shows an example of tetrahedron element.

“Tetrahedral elements are geometrically versatile and are suitable to be used in automatic meshing algorithms. As it has stiff edges, it is inherently rigid. For this reason, it is often used to stiffen frame structures. A difficulty with these elements is one of ordering of the nodal numbers and, in fact, of a suitable representation of a body divided into such elements.” (Zienkiewicz, Taylor, & Zhu, 2005).

“For tetrahedral elements, extremely fine meshes are required to obtain accurate results. This will result in very large numbers of simultaneous equations in practical problems. It may place a severe limitation on the use of the method in practice. In addition, the bandwidth of the resulting equation system becomes large. It increases the use of iterative solution methods.” (Zienkiewicz, Taylor, & Zhu, 2005).
Calculation steps for obtaining stiffness and force values for tetrahedron elements have been described below:

Consider a tetrahedron element with corner nodes numbered as 1, 2, 3 and 4. This numbering should be done in a specific order to avoid the calculation of negative volumes. The numbering system is such that when viewed from the last node (say, node 1), the first three nodes are numbered in a counterclockwise manner, such as 4, 3, 2, 1 or 3, 2, 4, 1.

![Figure 4.02: Four node tetrahedral solid element (Logan, 2007).](image)

Displacements along x, y and z axes are given by \( u, v \) and \( w \) respectively. The unknown nodal displacements are now given by equation 4.01. There are three degrees of freedom per node, or twelve total degrees of freedom per element.

\[
\{d\} = \begin{bmatrix}
  u_1 \\
  v_1 \\
  w_1 \\
  \vdots \\
  u_4 \\
  v_4 \\
  w_4 
\end{bmatrix}
\]

(4.01)
The linear displacement functions $u$, $v$ and $w$ are then selected as

\[
\begin{align*}
u(x, y, z) &= a_1 + a_2 x + a_3 y + a_4 z \\
v(x, y, z) &= a_5 + a_6 x + a_7 y + a_8 z \\
w(x, y, z) &= a_9 + a_{10} x + a_{11} y + a_{12} z
\end{align*}
\]  

(4.02)

The $a_i$'s can be expressed in terms of the known nodal coordinates $(x_1, y_1, z_1, \ldots, z_4)$ and the unknown nodal displacements $(u_1, v_1, w_1, \ldots, w_4)$ of the element. We obtain $u(x, y, z)$ vector as

\[
\begin{align*}
u(x, y, z) &= \frac{1}{6V} \left\{ (\alpha_1 + \beta_1 x + \gamma_1 y + \delta_1 z)u_1 + (\alpha_2 + \beta_2 x + \gamma_2 y + \delta_2 z)u_2 + (\alpha_3 + \beta_3 x + \gamma_3 y + \delta_3 z)u_3 + (\alpha_4 + \beta_4 x + \gamma_4 y + \delta_4 z)u_4 \right\}
\end{align*}
\]  

(4.03)

where $V$ is the volume of the tetrahedron and the coefficients $\alpha_i$, $\beta_i$, $\gamma_i$, and $\delta_i$ ($i = 1, 2, 3, 4$) in equation (4.03) are given by

\[
\begin{align*}
\alpha_1 &= \left| \begin{array}{ccc}
x_2 & y_2 & z_2 \\
x_3 & y_3 & z_3 \\
x_4 & y_4 & z_4
\end{array} \right|, & \beta_1 &= \left| \begin{array}{ccc}
1 & y_2 & z_2 \\
1 & y_3 & z_3 \\
1 & y_4 & z_4
\end{array} \right|, & \gamma_1 &= \left| \begin{array}{ccc}
x_2 & z_2 \\
x_3 & z_3 \\
x_4 & z_4
\end{array} \right|, & \delta_1 &= \left| \begin{array}{ccc}
x_2 & y_2 \\
x_3 & y_3 \\
x_4 & y_4
\end{array} \right|
\end{align*}
\]  

(4.04)

\[
\begin{align*}
\alpha_2 &= -\left| \begin{array}{ccc}
x_1 & y_1 & z_1 \\
x_3 & y_3 & z_3 \\
x_4 & y_4 & z_4
\end{array} \right|, & \beta_2 &= \left| \begin{array}{ccc}
y_1 & z_1 \\
y_3 & z_3 \\
y_4 & z_4
\end{array} \right|, & \gamma_2 &= -\left| \begin{array}{ccc}
x_1 & z_1 \\
x_3 & z_3 \\
x_4 & z_4
\end{array} \right|, & \delta_2 &= \left| \begin{array}{ccc}
x_1 & y_1 \\
x_3 & y_3 \\
x_4 & y_4
\end{array} \right|
\end{align*}
\]  

(4.05)

\[
\begin{align*}
\alpha_3 &= \left| \begin{array}{ccc}
x_1 & y_1 & z_1 \\
x_2 & y_2 & z_2 \\
x_4 & y_4 & z_4
\end{array} \right|, & \beta_3 &= -\left| \begin{array}{ccc}
y_1 & z_1 \\
y_2 & z_2 \\
y_4 & z_4
\end{array} \right|, & \gamma_3 &= \left| \begin{array}{ccc}
x_1 & z_1 \\
x_2 & z_2 \\
x_4 & z_4
\end{array} \right|, & \delta_3 &= -\left| \begin{array}{ccc}
x_1 & y_1 \\
x_2 & y_2 \\
x_4 & y_4
\end{array} \right|
\end{align*}
\]  

(4.06)
The vectors \( v(x,y,z) \) and \( w(x,y,z) \) are obtained by substituting \( v_i \)'s for all \( u_i \)'s and then \( w_i \)'s for all \( u_i \)'s in equation (4.03). These three displacement vectors can be written equivalently in expanded form in terms of the shape functions and unknown nodal displacements as

\[
\begin{bmatrix}
  u \\
  v \\
  w
\end{bmatrix} = \begin{bmatrix}
  N_1 & 0 & 0 & N_2 & 0 & 0 & N_3 & 0 & 0 & N_4 & 0 & 0 \\
  0 & N_1 & 0 & 0 & N_2 & 0 & 0 & N_3 & 0 & 0 & N_4 & 0 \\
  0 & 0 & N_1 & 0 & 0 & N_2 & 0 & 0 & N_3 & 0 & 0 & N_4
\end{bmatrix} \begin{bmatrix}
  u_1 \\
  v_1 \\
  w_1 \\
  \vdots \\
  u_4 \\
  v_4 \\
  w_4
\end{bmatrix}
\]

The 3 x 12 matrix on the right side of the equation (4.09) is called shape function matrix \([N]\).

The shape functions \( N_1, N_2, N_3, N_4 \) are given by equation (4.10).

\[
N_1 = \frac{(\alpha_1 + \beta_1 x + \gamma_1 y + \delta_1 z)}{6V} \quad \quad N_2 = \frac{(\alpha_2 + \beta_2 x + \gamma_2 y + \delta_2 z)}{6V}
\]
The element strains for the three-dimensional stress state are given by equation (4.11).

\[
\begin{align*}
N_3 &= \frac{(\alpha_3 + \beta_3 x + \gamma_3 y + \delta_3 z)}{6V} \\
N_4 &= \frac{(\alpha_4 + \beta_4 x + \gamma_4 y + \delta_4 z)}{6V}
\end{align*}
\] (4.10 b)

Using equation (4.09) in equation (4.11), we obtain

\[
[e] = [B][d]
\] (4.12)

where

\[
[B] = [B_1 \ B_2 \ B_3 \ B_4]
\] (4.13)

The sub-matrix \(B_i\) in equation (4.13) is defined by

\[
B_1 = \begin{bmatrix}
N_{1x} & 0 & 0 \\
0 & N_{1y} & 0 \\
0 & 0 & N_{1z} \\
N_{1y} & N_{1x} & 0 \\
0 & N_{1z} & N_{1y} \\
N_{1z} & 0 & N_{1x}
\end{bmatrix}
\] (4.14)
where $N_{1,x}$, $N_{1,y}$, and $N_{1,z}$ are the differentiation of shape function $N_i$ with respect to the variable $x$, $y$ and $z$ respectively. Sub-matrices $B_2$, $B_3$, and $B_4$ are defined by simply indexing the subscript in equation (4.14) from 1 to 2, 3, and then 4, respectively. Substituting the shape functions from equation (4.10) into equation (4.14), $B_i$ can be calculated as

$$B_i = \frac{1}{6V} \begin{bmatrix}
\beta_i & 0 & 0 \\
0 & \gamma_i & 0 \\
0 & 0 & \delta_i \\
\gamma_i & \beta_i & 0 \\
0 & \delta_i & \gamma_i \\
\delta_i & 0 & \beta_i
\end{bmatrix}$$

(4.15)

Similarly, we can calculate the sub-matrices $B_2$, $B_3$, and $B_4$.

The relation between element stresses and the element strains is expressed by

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

(4.16)

Where the constitutive matrix $[D]$ for an elastic material is now given by

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

(4.17)

The element stiffness matrix can be written as
For a simple tetrahedron element, both matrices \([B]\) and \([D]\) are constant. So equation (4.18) can be simplified to

\[
[k] = [B]^{T} [D] [B] V
\]  

(4.19)

where, \(V\) is the volume of the element. The element stiffness matrix is now a 12x12 matrix.

The element body force matrix is given by equation (4.20).

\[
\{f_b\} = \int_{V} \{N\}^{T} \{X\} dV
\]  

(4.20)

Where \([N]\) is given by the 3 x 12 matrix in equation (4.09), and

\[
\{X\} = \begin{bmatrix} X_3 \\ Y_3 \\ Z_3 \end{bmatrix}
\]  

(4.21)

Assuming that the element's body force is constant, the nodal components of body forces can be calculated as one fourth of the total resultant body force. That is,

\[
\{f_b\}_i = \frac{1}{4} \begin{bmatrix} X_3 & Y_3 & Z_3 & X_3 & Y_3 & Z_3 & X_3 & Y_3 & Z_3 \end{bmatrix}^{T}
\]  

(4.22)

The element body force is then a 12 x 1 matrix.

The surface forces are given by equation (4.23).
where $[N]$ is the shape function matrix evaluated on the surface where the surface traction occurs.

Let $p$ an uniform pressure acting on the surface with nodes 1-2-3 of the tetrahedron element shown in figure 4.01. The $x$, $y$ and $z$ components of $p$ are $p_x$, $p_y$, and $p_z$ respectively. The resulting nodal forces can be calculated as

$$
\{f_2\} = \int_\Sigma \left[ N_z \right]^T \{ T \} dS
$$

(4.23)

Simplifying and integrating equation (4.24), can show that

$$
\{f_2\} = \int_\Sigma \left[ N_z \right]^T \text{evaluated on surface,2,3} \begin{bmatrix} p_x \\ p_y \\ p_z \end{bmatrix} dS
$$

(4.24)

where $S_{123}$ is the surface area of the element's surface associated with nodes 1-2-3 (Logan, 2007).
Many structural analysis problems cannot be solved analytically, but very good approximate solutions can be obtained by finite element methods for those problems. Yet in using finite element method there are some conditions for which problems arise. Analysis problems where nearly incompressible materials are involved is one of those conditions.

Compressibility and incompressibility of a material is measured by Poisson ratio, $\nu$. When a material is compressed in one direction, it gets expanded in the direction perpendicular to the direction of compression. Ratio of this lateral change in length to the change in longitudinal length is known as Poisson ratio.

Poisson's ratio can be expressed as

$$\nu = -\frac{\varepsilon_{\text{tran}}}{\varepsilon_{\text{long}}} \quad (5.01)$$

where

- $\nu = \text{Poisson's ratio}$
- $\varepsilon_{\text{tran}} = \text{transverse strain}$
- $\varepsilon_{\text{long}} = \text{longitudinal strain}$

The Poisson's ratio is in the range 0 - 0.5 for most common materials. Materials, for which Poisson ratio is very close to 0.5, are categorized as nearly incompressible materials.
For nearly incompressible materials, finite element methods give worthless results. Stiffness for this kind of materials is much greater than would be expected. This problem is known as *locking situation*. A study on locking problem titled as ‘Introduction to Locking in Finite Element Method’ was conducted by *W A M Brekelmans* to show the stiffness values for different materials against different Poisson’s ratio. He has developed a *Stiffness vs Poisson’s ratio chart* as shown in figure 5.01. This figure shows that for nearly incompressible materials (Poisson ratio nearly about 0.5) stiffness become abnormally high.

![Stiffness on ri vs Poisson's ratio](image)

**Figure 5.01:** Material’s Stiffness vs its Poisson ratio (Brekelmans & Den, 2005).

The volumetric strain for nearly incompressible material is nearly zero as its stiffness is very high. For this reason, it is not possible to calculate the sum of normal stresses for nearly incompressible material accurately by finite element method. The error gets increased when direct computation of the sum of the normal stresses engages multiplication of the volumetric
strain by a large number, the bulk modulus. Therefore, it is required to calculate the stresses in nearly incompressible material indirectly.

Szabo et al. have described an indirect computation technique to calculate the stresses in nearly incompressible materials in a research report titled as “Stress Computation for Nearly Incompressible Materials”. The calculation process presented in that research paper has been summarized here:

Let first consider a domain $\Omega$ of elastic material of constant thickness $t_x$. Displacements are denoted by $u_x$ and $u_y$. Corresponding strain components are expressed by $\varepsilon_x$, $\varepsilon_y$ and $\gamma_{xy}$.

By definition:

$$
\varepsilon_x = \frac{\partial u_x}{\partial x}, \quad \varepsilon_y = \frac{\partial u_y}{\partial y}, \quad \gamma_{xy} = \frac{\partial u_x}{\partial y} + \frac{\partial u_y}{\partial x} \quad (5.02)
$$

Relation between stress and strain is given by Hook’s law:

$$
\sigma_x = \lambda(\varepsilon_x + \varepsilon_y) + 2G\varepsilon_x \quad (5.03a)
$$

$$
\sigma_y = \lambda(\varepsilon_x + \varepsilon_y) + 2G\varepsilon_y \quad (5.03b)
$$

$$
\sigma_x = \lambda(\varepsilon_x + \varepsilon_y) \quad (5.03c)
$$

$$
\tau_{xy} = G\gamma_{xy} \quad (5.03d)
$$

Where

$$
\lambda = \frac{E\nu}{(1+\nu)(1-2\nu)} \quad G = \frac{E}{2(1+\nu)} \quad (5.04)
$$

In matrix form we can write the equations (5.03) as:

$$
\{\sigma\} = [E]\{\varepsilon\} \quad (5.05a)
$$
Where

\[
\{\sigma\} = \begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} \quad \{\varepsilon\} = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix}
\]  

(5.05b)

And

\[
[E] = \begin{bmatrix} \lambda + 2G & \lambda & 0 \\ \lambda & \lambda + 2G & 0 \\ 0 & 0 & G \end{bmatrix}
\]  

(5.05c)

$[E]$ can be written as:

\[
[E] = [T]^T[D][T]
\]  

(5.06a)

Where $[D]$ is the diagonal matrix of the eigenvalues of $[E]$ and $[T]$ is the matrix of normalized eigenvectors of $[E]$. Specifically, $[D]$ and $[T]$ are:

\[
[D] = \begin{bmatrix} 2(\lambda + G) & 0 & 0 \\ 0 & 2G & 0 \\ 0 & 0 & G \end{bmatrix}
\]  

(5.06b)

\[
[T] = \begin{bmatrix} 1/\sqrt{2} & 1/\sqrt{2} & 0 \\ 1/\sqrt{2} & -1/\sqrt{2} & 0 \\ 0 & 0 & 1 \end{bmatrix}
\]  

(5.06c)

The strain energy is:

\[
U(\ddot{u}) = \frac{1}{2} \iint_{\Omega} (\sigma_x \varepsilon_x + \sigma_y \varepsilon_y + \tau_{xy} \gamma_{xy}) \, dx \, dy
\]

(5.07)

\[
= \frac{1}{2} \iint_{\Omega} \{\varepsilon\}^T [E] \{\varepsilon\}
\]

The energy norm of $\ddot{u}$, which is required to error calculation, is denoted by $\|\ddot{u}\|_E$. By definition:

\[
\|\ddot{u}\|_E = \sqrt{U(\ddot{u})}
\]  

(5.08)

Using equations (5.06) a, b and c it can be written:
\[ U(\ddot{u}) = \frac{1}{2} \int_{\Omega} \left[ \dddot{\lambda} + G \left( \varepsilon_x^{(u)} + \varepsilon_y^{(u)} \right)^2 + G \left( \varepsilon_x^{(u)} - \varepsilon_y^{(u)} \right)^2 \right] \, dx \, dy \]  

(5.09)

Let the exact solution is \( \ddot{u}_{\text{Ex}} \). The domain \( \Omega \) is required to subdivide in \( M \) numbers of mesh elements to get the finite element approximation. Cartesian components \( x, y \) can be expressed as functions of \( \xi \) and \( \eta \). The functions are:

\[
\begin{align*}
    x &= f_x^i(\xi, \eta), \\
y &= f_y^i(\xi, \eta), \\
    \xi, \eta &\in \Omega, \quad \text{where } i = 1, 2, 3, \ldots, M
\end{align*}
\]  

(5.10)

Set of all functions is denoted by \( S(\Omega, M, f) \) or simply \( S \). \( \bar{S} \) are the admissible functions in \( S \) which satisfy the kinematic boundary conditions. The dimension of \( \bar{S} \) is the number of degrees of freedom, \( N \).

The finite element solution \( \ddot{u}_{\text{FE}} \) is that function from \( \bar{S} \) which minimizes the strain energy of the error:

\[ U(\ddot{u}_{\text{EX}} - \ddot{u}_{\text{FE}}) = \min_{\ddot{u} \in \bar{S}} U(\ddot{u}_{\text{EX}} - \ddot{u}) \]  

(5.11)

Expressing \( \ddot{e} = \ddot{u}_{\text{EX}} - \ddot{u}_{\text{FE}} \), it can be written

\[ U(\ddot{e}) = \frac{1}{2} \int_{\Omega} \left[ \dddot{\lambda} + G \left( \varepsilon_x^{(e)} + \varepsilon_y^{(e)} \right)^2 + G \left( \varepsilon_x^{(e)} - \varepsilon_y^{(e)} \right)^2 \right] \, dx \, dy \]  

(5.12)

The error in the strain energy \( U(\ddot{e}) \) can be reduced by

1. Successive mesh refinement which known as h-extension, or by
2. Increasing the polynomial degree of elements which is known as p-extension, or by
3. Combined process of both which is known as hp-extension.

The p-extension process is independent of \( \nu \), while h-extension process is substantially dependant on \( \nu \). That is why nearly incompressible materials are analyzed by p-extension error reduction process.
The root-mean-square stress $S(\bar{u})$ as per definition:

$$S(\bar{u}) = \sqrt{\frac{1}{V} \int_{\Omega} \left[ (\sigma_{x}^{(u)})^2 + (\sigma_{y}^{(u)})^2 + (\tau_{xy}^{(u)})^2 \right] dxdy}$$  \hspace{1cm} (5.13)$$

where $V$ is the volume.

Using equations (5.05a), (5.05b), (5.05c) and (5.06a) $S^2(\bar{u})$ can be expressed in terms of strains.

$$S^2(\bar{u}) = \frac{1}{V} \int_{\Omega} \left[ 2(\lambda + G)^2 (\varepsilon_{x}^{(u)} + \varepsilon_{y}^{(u)})^2 + 2G^2 (\varepsilon_{x}^{(u)} - \varepsilon_{y}^{(u)})^2 + G^2 (\gamma_{xy}^{(u)})^2 \right] dxdy$$  \hspace{1cm} (5.14)$$

This equation for $S^2(\bar{u})$ is similar to the equation (5.09) for calculating $U(\bar{u})$. So we can express the square of the error $S^2(\bar{e})$ similar to the equation for $U(\bar{e})$. The square of the error in the sum of the normal stresses integrated over the volume is:

$$S^2_1(\bar{e}) = \frac{1}{V} \int_{\Omega} 2(\lambda + G)^2 (\varepsilon_{x}^{(e)} + \varepsilon_{y}^{(e)})^2 t_{x} dxdy = \frac{1}{2V} \int_{\Omega} (\sigma_{x}^{(e)} + \sigma_{y}^{(e)})^2 t_{x} dxdy$$  \hspace{1cm} (5.15a)$$

$$\leq \frac{1}{V} (\lambda + G) U(\bar{\varepsilon})$$  \hspace{1cm} (5.15b)$$

The constant $\lambda$ in the equation (5.15b) depend on Poisson’s ratio. As the ratio becomes close to 0.5, result becomes infinity. On the other hand, the error in the differences of normal stresses and shear stress is very small. It is because Poisson’s ratio has no effect on those error estimations.

$$S^2_2(\bar{e}) = \frac{1}{V} \int_{\Omega} 2G^2 (\varepsilon_{x}^{(e)} - \varepsilon_{y}^{(e)})^2 t_{x} dxdy = \frac{1}{2V} \int_{\Omega} (\sigma_{x}^{(e)} - \sigma_{y}^{(e)})^2 t_{x} dxdy \leq \frac{4}{V} G U(\bar{\varepsilon})$$  \hspace{1cm} (5.16)$$

$$S^2_3(\bar{e}) = \frac{1}{V} \int_{\Omega} G^2 (\gamma_{xy}^{(e)})^2 t_{x} dxdy = \frac{1}{V} \int_{\Omega} (\gamma_{xy}^{(e)})^2 t_{x} dxdy \leq \frac{2}{V} G U(\bar{\varepsilon})$$  \hspace{1cm} (5.17)$$
Equations (5.15), (5.16) and (5.17) indicate that good approximations can be obtained for 
\( \sigma_x - \sigma_y \) and \( \gamma_{xy} \) but not for \( \sigma_x + \sigma_y \).

However, the sum of normal stresses satisfies the Laplace’s equation when body forces are constant (or zero).

\[ \Delta(\sigma_x + \sigma_y) = 0 \]  \hspace{1cm} (5.18)

Considering the above fact and the fact that sum of normal stresses is invariant with respect to coordinate transformation, we can write:

\[ \sigma_x + \sigma_y = \sigma_n + \sigma_t \]  \hspace{1cm} (5.19)

Where \( \sigma_n \) and \( \sigma_t \) are the stresses in positive normal direction and positive tangent direction of any boundary segments respectively. These boundary conditions can be calculated accurately using the equations for \( \sigma_x^{uFE} - \sigma_y^{uFE} \) and \( \gamma_{xy}^{uFE} \). Equation (5.19) can be written as follow:

\[ \sigma_x + \sigma_y = \sigma_n + \sigma_t = (\sigma_n^{uFE} - \sigma_t^{uFE}) + 2\sigma_n \]  \hspace{1cm} (5.20)

This is how we can calculate stresses for nearly incompressible materials (Szabo, Babuska, & Chayapathy, 1988).
CHAPTER 06
LUMPED PLASTICITY MODEL OF FRAME ELEMENT

Usually structures are designed to response elastically to small magnitude earthquakes. In high seismic regions, structures will not response elastically to the maximum magnitude of an earthquake. Response of structures to high magnitude earthquake is nonlinear. Nonlinearity of a frame element is illustrated in terms of moment-rotation relation. There are mainly two types of material nonlinearity in frame elements. Those are lumped plasticity and distributed plasticity. Distributed plasticity model is used to obtain more precise assessment of the nonlinear structural response. In a contrary, the lumped plasticity model is extensively used as because it is simple to formulate. In this model, a frame element represented by an elastic beam-column element connecting two zero length nonlinear rotational spring elements. Frame element forms plastic hinges at the end node region when its response is nonlinear.

Inelastic deformation gradually spread out into the member as a function of loading history. In the lumped plasticity model, actual behavior of inelastic deformation is simplified. This simplification helps to reduce computational cost and storage requirements. However, this simplification ignores some significant features of the hysteretic actions of the structure. That is why there are some limitations to apply this model. In addition, the lumped plasticity model is not capable to describe adequately the deformation softening behavior of reinforced concrete members. More advanced models are needed to observe such deformation softening (Filippou & Spacone, 1991).

To formulate a frame element let us consider a three-dimensional frame element with plastic hinges at end sections. Again, consider that an end section is bounded by flat surfaces as shown in
As per their suggestion a satisfactory representation of the axial load–biaxial bending moment bounding surface of the elastic domain can be obtained considering 26 flat surfaces. Out of those flat surfaces six surfaces are normal to the principal axes $x$, $y$ and $z$; twelve surfaces are normal to the bisections of the $y$-$z$, $x$-$y$ and $x$-$z$ principal planes; eight surfaces are normal to the bisections of octants.

Figure 6.01: Flat surfaces approximating the elastic domain for the end sections (Mazza & Mazza, 2012).

We have considered that the element is subjected to biaxial bending with axial force. There are two components of the proposed Lumped Plasticity Model required to analyze the element. One is elastic-perfectly plastic represented by a bilinear moment-curvature ($M$-$\phi$) law, another is linearly elastic characterized by the flexural stiffness $\rho EI$. Here $\rho$ is the hardening ratio.
It is assumed that the inelastic deformation is lumped at the cross section. This deformation is represented by the axial strain $\varepsilon_p$ along the longitudinal axis $x$, and the curvature $\phi_{py}$ and $\phi_{pz}$ along the principal axes $y$ and $z$. We can write,

$$\varepsilon_p = \left[\varepsilon_p, \phi_{py}, \phi_{pz}\right]^T$$

(6.01)

Corresponding stresses are denoted by

$$\sigma = \left[N, M_y, M_z\right]^T$$

(6.02)

Moreover, corresponding plastic stresses are denoted by $\sigma_{pk}$ related to $n_k$. Here $n_k$ indicates the ‘$k$th normal direction’. For each normal direction, the elastic domain $g(\sigma)=0$ can be approximated by $n_{k0}$ flat surfaces as $g_k(\sigma)$. The piecewise linearized elastic domain is characterized by the $N$ matrix.

$$N = \begin{bmatrix}
1 & -1 & 0 & 0 & 0 & 0 & 0 & 0 & c_{yx} & c_{yx} & -c_{yx} \\
0 & 0 & 1 & -1 & 0 & 0 & 1 & -1 & 1 & 1 & -1 & 1 \\
0 & 0 & 0 & 0 & 1 & -1 & c_{yz} & c_{yz} & -c_{yz} & -c_{yz} & 0 & 0 & 0 \\
-c_{yx} & -c_{zx} & c_{zx} & -c_{zx} & c'_{yz} & c'_{yz} & -c'_{yz} & -c'_{yz} & c'_{yz} & c'_{yz} & -c'_{yz} & -c'_{yz} & \end{bmatrix}$$

(6.03)

Where

$$c_{yx} = \frac{\|\sigma_{p5} - \sigma_{p4}\|}{\|\sigma_{p1} - \sigma_{p2}\|}$$

$$c_{zx} = \frac{\|\sigma_{p5} - \sigma_{p6}\|}{\|\sigma_{p1} - \sigma_{p2}\|}$$
\[ c_{yz} = \cos \psi \frac{\sigma_3 - \sigma_4}{\sigma_5 - \sigma_6} \quad c_{yz}' = \cos \psi \frac{\sigma_7 - \sigma_{10}}{\sigma_1 - \sigma_2} \]  
(6.04)

The generalized stresses considered in equations (6.04) are,

\[
\begin{align*}
\sigma_{p1} &= \begin{bmatrix} N_{p1} \\ 0 \\ 0 \end{bmatrix}, \quad \sigma_{p2} = \begin{bmatrix} N_{p2} \\ 0 \\ 0 \end{bmatrix}, \quad \sigma_{p3} = \begin{bmatrix} N_{p3} \\ M_{p7} \end{bmatrix}, \quad \sigma_{p4} = \begin{bmatrix} N_{p4} \\ M_{p8} \end{bmatrix}, \quad \sigma_{p5} = \begin{bmatrix} 0 \\ M_{p5} \end{bmatrix} \\
\sigma_{p6} &= \begin{bmatrix} N_{p6} \\ 0 \\ M_{p7} \end{bmatrix}, \quad \sigma_{p7} = \begin{bmatrix} N_{p7} \\ M_{p8} \end{bmatrix}, \quad \sigma_{p8} = \begin{bmatrix} N_{p8} \\ M_{p9} \end{bmatrix}, \quad \sigma_{p9} = \begin{bmatrix} 0 \\ M_{p10} \end{bmatrix}, \quad \sigma_{p10} = \begin{bmatrix} N_{p10} \\ 0 \end{bmatrix} 
\end{align*}
\]  
(6.05)

The components of the generalized plastic stress vector \( \sigma_{pk} \) can be written as:

\[
\begin{align*}
N_{pk} &= \int_{A_c} \sigma_c dA + \sum_{i=1}^{n_b} A_{si} \sigma_{si} \\
M_{pyk} &= -\int_{A_c} \sigma_c zdA - \sum_{i=1}^{n_b} A_{si} \sigma_{si} z_i \\
M_{pzk} &= \int_{A_c} \sigma_c ydA - \sum_{i=1}^{n_b} A_{si} \sigma_{si} y_i 
\end{align*}
\]  
(6.06, 6.07, 6.08)

In above equations, \( A_c \) = area of compressed concrete section

\( n_b \) = number of longitudinal bars in the structural element

\( A_{si} \) = cross sectional area of each bar

\((y_i, z_i)\) denotes the position
This nonlinear analysis is an iterative procedure. Once the initial state and the incremental load is known the elastic-plastic response of the structural element is determined by the Haar-Karman principle. According to this principal, among all the generalized stress fields $\sigma$ satisfying equilibrium, the elastic-plastic solution $\sigma_{EP}$ is the stress field which has minimum distance. Distance is expressed in terms of complementary $\lambda_c$ energy, which is

$$\lambda_c(\sigma_{EP}) = \frac{L}{2} \int_0^1 (\sigma_{EP} - \sigma_E)^T D_c^{-1}(\sigma_{EP} - \sigma_E) d\xi = \min$$

(6.09)

Where $L$ = the length of the element, $\xi = x/L$ and $D_c$ is the elastic matrix. In addition, we have to satisfy the condition

$$g_k(\sigma_{EP}) \leq 0 \quad \text{for } k = 1 \ldots n_{fs}$$

(6.10)

The elastic-plastic solution is determined from the tangent point between the energy level curve and the bounding surface of the elastic domain (Mazza & Mazza, 2012).
WORKS CITED


