SOLUTION OF TRUSS PROBLEMS BY USING FINE ELEMENT METHOD

Usman.A.¹, N.A.Shahid¹, M.F.Tabassum², A. Sana¹, N. A. Chaudhry³, M.Saeed²
¹Department of Mathematics, Lahore Garrison University, Lahore, Pakistan.
²Department of Mathematics, University of Management and Technology, Lahore, Pakistan.
³Department of Mathematics, UCEST, Lahore Leads University, Lahore, Pakistan

Abstract: The Direct Stiffness Method (DSM) is the most straight forward technique for assembling the system matrices required for Finite Element Method (FEM). In this paper, DSM has been used to find displacement components of nodes, the reaction force components at different nodes, element displacements and strains and stesses of element in truss structures. As a comprehensive example of two dimensional truss analyses, the structure is analyzed to obtain displacement, reaction forces, strains and stesses using FEM. All the calculations are done manually and checked by using MATLAB programming.

Keywords: Finite Element Method, Direct Stiffness Method, Structural Analysis, Two Dimensional Truss Analysis.

1. INTRODUCTION

The mathematical root of the finite element method goes back to the history at least a half century. Approximate methods for solving differential equations using trial solutions are even older in origin. Lord Rayleigh and Ritz [1] used trial functions to approximate solutions of differential equations. Galerkin [2] used the same concept for solution. The deficiency in the past approaches [6] as compared to the present finite element method, is that the trial functions have to apply over the whole domain of the related problem, while the Galerkin method provides a very highly approach for the finite element method.

In modern ages between 1980 and 2000, the work on finite element method has been increased for the implementation in pressure vessels, shell bending and basic three dimension situations in elastic structural analysis [4,5] with also fluid flow and heat transfer [7]. More expedition of FEM is in deflection and dynamic structure [8] that has been also represented during these decades.

The term displacement is comparatively general in the finite element method and can represent for example, physical displacement, temperature and fluid velocity. First of all this term was used by Clough in [3] in the topic of plane stress analysis and it has frequently used since current days.

2. MATERIAL AND METHODS

2.1 Finite Element Analysis

When external loads are applied to a system then stiffness matrix is used to calculate the relation between loads and displacements. We can get strain and stress of each element, which we want after using backward substitution of displacements into each element equations. This technique is called DSM by using FEM.

There is another scheme named Flexibility Method [17] in FEM. These problems are non-structural in which ‘displacements’ are given as ‘quantities’ and ‘forces’ are given as ‘variables’.

If we find stiffness matrix in the direct stiffness method that has quality of every element which is changed from the element coordinates system to the global coordinate system. First we find element stiffness matrix of every changed element then element values are directly substituted to the global stiffness matrix. This concept helps to make the element transformation and stiffness matrix assembly procedure.

When we operate direct stiffness matrix in FEM, we see that global node is mathematically dull.

Finally, we can use another signification in place of stiffness method, is displacement compatibility [18-20]. It also uses to make procedure. It is assured that algebraic back tracking is used to find strain and stress.

2.2 Nodal Equilibrium Equations

First we make element equations by using element coordinates to global coordinates and assembly of the global equilibrium equations in the two dimensions. A simple two dimension truss made of two structural members converted with pin and with condition, external forces will be applied. The connection of pin holds at node and element numbers with global coordinate system.

Symbolically, we use \( U_{2i} \) as a global displacement. The sense of \( U_{2i-1} \) and \( U_{2i} \) is that \( U_{2i-1} \) is displacement of global X-direction of node \( i \) and \( U_{2i} \) is displacement of global Y-direction of node \( i \). We use odd and even numbered for the displacements in the direction of the global X-axis and Y-axis respectively.

Comparing vector components of element displacements to global displacements, we have

\[
\begin{align*}
\delta_1^{(e)} &= U_1^{(e)} \cos \theta + U_2^{(e)} \sin \theta \\
\delta_2^{(e)} &= U_1^{(e)} \sin \theta + U_2^{(e)} \cos \theta
\end{align*}
\]

(1)

We know that component \( v \) displacement is not related to stiffness of element, so it will also not be related to element forces. Thus, the axial deformation of the element becomes as

\[
\delta^{(e)} = U_1^{(e)} - U_1^{(e)} = [U_1^{(e)} - U_1^{(e)}] \cos \theta + [U_2^{(e)} - U_2^{(e)}] \sin \theta
\]

(2)

Net axial force operating on the individual becomes as

\[
F^{(e)} = k^{(e)} \delta^{(e)} = k^{(e)} [U_1^{(e)} - U_1^{(e)}] \cos \theta + [U_2^{(e)} - U_2^{(e)}] \sin \theta
\]

(3)

May-June
The equilibrium relation between two element trusses is

\[-k^{(1)}(U_5 - U_1)\cos\theta_1 + (U_6 - U_2)\sin\theta_1 \cos\theta_1 = F_1\]

\[-k^{(2)}(U_5 - U_1)\cos\theta_1 + (U_6 - U_2)\sin\theta_1 \sin\theta_1 = F_2\]

\[-k^{(3)}(U_5 - U_1)\cos\theta_2 + (U_6 - U_4)\sin\theta_2 \cos\theta_2 = F_3\]

\[-k^{(4)}(U_5 - U_1)\cos\theta_2 + (U_6 - U_4)\sin\theta_2 \sin\theta_2 = F_4\]

\[-k^{(5)}(U_5 - U_3)\cos\theta_3 + (U_6 - U_4)\sin\theta_3 \cos\theta_3 = F_5\]

\[-k^{(6)}(U_5 - U_3)\cos\theta_3 + (U_6 - U_4)\sin\theta_3 \sin\theta_3 = F_6\]

We can summarize this equilibrium system as

\[ [K][U] = [F] \]

where

- \([K]\) = Global stiffness matrix
- \([U]\) = Nodal displacement vector
- \([F]\) = Nodal force vector

### 2.3 Transformation of Element

A direct method is a process which is used to find-out the essential properties on an element by element base. The barelement equation then expressed as

\[
\frac{AE}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} u_{x}^{(1)} \\ u_{x}^{(2)} \end{bmatrix} = k_s \begin{bmatrix} k_s & -k_s \\ -k_s & k_s \end{bmatrix} \begin{bmatrix} f_{x}^{(1)} \\ f_{x}^{(2)} \end{bmatrix}
\]

Global coordinates have the relation of element displacement and element axial displacement of coordinate system.

We use symbolically \(c = \cos\theta\), \(s = \sin\theta\). By applying matrix multiplications on R.H.S of above equation, it becomes:

\[
[K'] = k_s \begin{bmatrix} c^2 & sc & -c^2 & -sc \\ sc & s^2 & -sc & -s^2 \\ -c^2 & -sc & c^2 & sc \\ -sc & -s^2 & sc & s^2 \end{bmatrix}
\]

Where \(k_s = \frac{EA}{L}\) = Characteristic axial stiffness of any element.

Its determinant should be zero because we know that stiffness matrix will remain singular after transformation.

### 2.4 Direction Cosines

A finite element model can be designed by representing nodes at specified coordinate system after the definition of element having nodes and connected by each element.

Length of element in general form is

\[L = \sqrt{(X_j - X_i)^2 + (Y_j - Y_i)^2} \]

where nodes \(i\) and \(j\) are represented as \((X_i, Y_i)\) and \((X_j, Y_j)\) in the global coordinates. Now we represent unit vector as

\[ \lambda = \frac{1}{L}[X_j - X_i, Y_j - Y_i] = \cos\theta_i I + \cos\theta_j J \]

\(i\) and \(j\) are called unit vectors in the global coordinates directions of \(X\) and \(Y\) respectively.

The direction cosines are written as

\[
\cos\theta = \cos\theta_1 = \frac{X_j - X_i}{L} \\
\sin\theta = \sin\theta_1 = \frac{Y_j - Y_i}{L}
\]

### 2.5 Direct Assembly of Global Stiffness Matrix

The formulation of element stiffness matrix in global framework can be obtained by using equation (5) as:

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

Similarly for element \(2\) we are changing superscript. Elements displacement location vectors for the truss of Figure 1 is

Location vector for element-1: \(L_1 = [1, 2, 5, 6]\)

Location vector for element-2: \(L_2 = [3, 4, 5, 6]\)

### 2.6 Boundary Conditions, Constraint Forces

After getting the global stiffness matrix with the help of equilibrium equation or the direct stiffness method, the global displacements and applied forces for the Fig. 1, is of the form

\[
[K][U] = [F]
\]

We cannot find direct unique solution of global stiffness matrix because this is a singular matrix. In order to develop such type of equations, we are unable to take into account the constraint fixed on system displacements by the help of condition to disqualify rigid body motion. For this purpose, we can use displacements boundary condition as

\[U_1 = U_2 = U_3 = U_4 = 0\]

Only \(U_5\) and \(U_6\) displacements are left for taking continue process. Applying this boundary condition on equation (4), we get

\[
K_{15}U_5 + K_{16}U_6 = F_1 \\
K_{25}U_5 + K_{26}U_6 = F_2 \\
K_{35}U_5 + K_{36}U_6 = F_3 \\
K_{45}U_5 + K_{46}U_6 = F_4
\]

Above system has been reduced. In this reduced system \(F_1, F_2, F_3\) and \(F_4\) are the reaction forces on nodes 1 and 2. On the other hand, \(F_5\) and \(F_6\) are applied external forces of global. For finding \(U_5\) and \(U_6\) we will use external force components, to find these components values, we will solve the last two of equations (6).
This is a more general approach to find boundary conditions and evaluation of forces. If we use subscript $c$ on constrained displacements and $a$ on active (unconstrained) displacements. The above system of equations can be reduced as

$$
\begin{bmatrix}
K_{cc} & K_{ca} \\
K_{ac} & K_{aa}
\end{bmatrix}
\begin{bmatrix}
U_c \\
U_a
\end{bmatrix} = \begin{bmatrix}
F_c \\
F_a
\end{bmatrix}
$$

Here, $U_c$ values are given and follow them $F_c$. Since $U_a$ values are not given and we have to find by using sub-reduction as

$$
\begin{bmatrix}
K_{ac} \\
K_{ac}
\end{bmatrix} [U_c] + [K_{aa}] [U_a] = \{F_a\}
$$

$$
[U_a] = [K_{aa}]^{-1} ([F_a] - [K_{aa}] [U_c])
$$

(7)

We can use in a truss structure and we have assumed before that $\{U_c\}$ should not be zero. After finding the values of displacement from equation (7) then applying these displacement values, we have the following reaction forces system as

$$
\{F_c\} = [K_{cc}] [U_c] + [K_{ca}] [U_a]
$$

(8)

By the symmetry of the stiffness matrix, we can write,

$$
[K_{cc}] = [K_{ac}]^T
$$

2.7 Strain and Stress in an Element

The concept of the strain and stress in global displacements system are the final evaluation to the solution of the truss problem by using finite element method. For connecting nodes $i$ and $j$, the element displacements in the global coordinates are represented as

$$
\begin{align*}
\varepsilon_i &= U_i^{(e)} \cos \theta + U_2^{(e)} \sin \theta \\
\varepsilon_j &= U_3^{(e)} \cos \theta + U_4^{(e)} \sin \theta
\end{align*}
$$

Now we can find element axial strain by above equation as

$$
\varepsilon^{(e)} = \frac{d\varepsilon^{(e)}(x)}{dx} = \frac{d\varepsilon^{(e)}}{dx} \begin{bmatrix} N_1(x) & N_2(x) \end{bmatrix} \begin{bmatrix} u_1^{(e)} \\
u_2^{(e)}
\end{bmatrix}
$$

$$
= \left[ \frac{-1}{L_e} \right] \begin{bmatrix} u_1^{(e)} \\
u_2^{(e)} \end{bmatrix} - \begin{bmatrix} u_1^{(e)} \\
u_2^{(e)} \end{bmatrix}
$$

where $L_e$ represents element length. Also, we can find axial stress as

$$
\sigma^{(e)} = E\varepsilon^{(e)}
$$

We can find the element displacements by using the global but its converse does not hold yet. Therefore, the element strain according to global displacements is as

$$
\varepsilon^{(e)} = \frac{d\varepsilon^{(e)}(x)}{dx} = \frac{d\varepsilon^{(e)}}{dx} \begin{bmatrix} N_1(x) & N_2(x) \end{bmatrix} \begin{bmatrix} U_1^{(e)} \\
U_2^{(e)} \\
U_3^{(e)} \\
U_4^{(e)}
\end{bmatrix}
$$

Here, the transformation matrix of element is denoted by $[R]$. And the element stress according to global displacements is

$$
\sigma^{(e)} = E\varepsilon^{(e)} = E\frac{d\varepsilon^{(e)}(x)}{dx} = E\frac{d\varepsilon^{(e)}}{dx} \begin{bmatrix} N_1(x) & N_2(x) \end{bmatrix} \begin{bmatrix} U_1^{(e)} \\
U_2^{(e)} \\
U_3^{(e)} \\
U_4^{(e)}
\end{bmatrix}
$$

The element is in tension means the value of stress is positive and compression holds when its value is negative.

Model Problem DISCRIPTION

As a comprehensive example of two dimensional truss analyses, the structure shown in Fig. 2 is analyzed to obtain displacements, reaction forces, strains, and stresses. While we do not include all computational details, the example shows the required steps, in sequence, for a finite element analysis.

![Figure 2](image)

(a) Every individual element has $A = 1.5in^2$, $E = 10 \times 10^6 \text{psi}$. (b) Nodes, elements and global displacements.

we know that from equation (5)

$$
\begin{bmatrix}
K^{(i)}
\end{bmatrix} = k_e
$$

$$
\begin{bmatrix}
c^2 & sc & -c^2 & -sc \\
sc & s^2 & -sc & -s^2 \\
-c^2 & -sc & c^2 & sc \\
-sc & -s^2 & sc & s^2
\end{bmatrix}
$$

From Figure 2, we have

Length of elements is

$L_1 = L_3 = L_4 = L_5 = L_7 = L_8 = 40$

$L_2 = L_6 = 40\sqrt{2}$

Characteristics equations of elements are

$k_1 = k_3 = k_4 = k_5 = k_7 = k_8 = 3.75 \times 10^5 \text{lb/in}$

$k_2 = k_6 = 2.65 \times 10^5 \text{lb/in}$

The nodal coordinates are

$\theta_1 = \theta_3 = \theta_5 = \theta_7 = 0 \cdot \theta_4 = \theta_6 = \frac{\pi}{2}, \theta_8 = \frac{\pi}{4}$

$\theta_5 = \frac{\pi}{2} \cdot \frac{\pi}{4} = \frac{3\pi}{4}$

For Elements-1, 3, 5 and 7:
\[
\begin{bmatrix}
K^{(1)}
\end{bmatrix} = \begin{bmatrix}
K^{(2)}
\end{bmatrix} = \begin{bmatrix}
K^{(3)}
\end{bmatrix} = \begin{bmatrix}
1 & 0 & -1 & 0 \\
0 & 0 & 0 & 0 \\
-1 & 0 & 1 & 0 \\
0 & 0 & 0 & 0
\end{bmatrix} = 3.75 \times 10^5
\]

For elements 4 and 8:
\[
\begin{bmatrix}
K^{(4)}
\end{bmatrix} = \begin{bmatrix}
K^{(6)}
\end{bmatrix} = 3.75 \times 10^5
\]

For Element-2:
\[
\begin{bmatrix}
K^{(2)}
\end{bmatrix} = \begin{bmatrix}
\dfrac{2.65}{2} \times 10^5
\end{bmatrix}
\]

For Element-6:
\[
\begin{bmatrix}
K^{(6)}
\end{bmatrix} = \begin{bmatrix}
\dfrac{2.65}{2} \times 10^5
\end{bmatrix}
\]

Table 1: Global displacements according to elements.

<table>
<thead>
<tr>
<th>Global Displacement</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>2</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>0</td>
<td>2</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>3</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>4</td>
<td>0</td>
<td>2</td>
<td>2</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>7</td>
<td>0</td>
<td>3</td>
<td>3</td>
<td>3</td>
<td>0</td>
<td>3</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>0</td>
<td>4</td>
<td>4</td>
<td>4</td>
<td>0</td>
<td>4</td>
<td>2</td>
</tr>
<tr>
<td></td>
<td>9</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>3</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>4</td>
<td>2</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>11</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>12</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>4</td>
<td>4</td>
</tr>
</tbody>
</table>

Location vector for each element and element node connectivity table.

Location vector for element-1: \(L_a = [1 \ 2 \ 5 \ 6] \)

Location vector for element-2: \(L_a = [1 \ 2 \ 7 \ 8] \)

Location vector for element-3: \(L_a = [3 \ 4 \ 7 \ 8] \)

Location vector for element-4: \(L_a = [5 \ 6 \ 7 \ 8] \)

Location vector for element-5: \(L_a = [5 \ 6 \ 9 \ 10] \)

Location vector for element-6: \(L_a = [9 \ 10 \ 7 \ 8] \)

Location vector for element-7: \(L_a = [7 \ 8 \ 11 \ 12] \)

Location vector for element-8: \(L_a = [9 \ 10 \ 11 \ 12] \)

Table 2: Relationship of connectivity between elements and nodes.

<table>
<thead>
<tr>
<th>Element</th>
<th>Node i</th>
<th>Node j</th>
<th>Element</th>
<th>Node i</th>
<th>Node j</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>3</td>
<td>5</td>
<td>3</td>
<td>5</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>4</td>
<td>6</td>
<td>5</td>
<td>4</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>4</td>
<td>7</td>
<td>4</td>
<td>6</td>
</tr>
</tbody>
</table>

Now we find the resulting components (individual terms) of the global stiffness matrix.

Node 1 and 2 are fixed. The displacement constraints \(U_1 = U_2 = U_3 = U_4 = 0\)

The global equilibrium equation

\[
\begin{bmatrix}
F_1 \\
F_2 \\
F_3 \\
F_4
\end{bmatrix} = \begin{bmatrix}
7.5 \\
3.75 \\
6.4 \\
0.75
\end{bmatrix}
\]

we use dash line for showing reaction forces and active displacements in prominent style. Also, they have been resolved into parts which are shown in the equation as follows

\[
\begin{bmatrix}
k_{xx} & k_{yx} \\
k_{xy} & k_{yy}
\end{bmatrix}\begin{bmatrix}
U_x \\
U_y
\end{bmatrix} = \begin{bmatrix}
F_x \\
F_y
\end{bmatrix}
\]

Solving above system of equations governing the active displacement, we have

\[
\begin{align*}
U_x &= 0.02133 \\
U_y &= 0.04085 \\
F_x &= 0.04169 \\
F_y &= 0.04069
\end{align*}
\]

Now we can find all the reaction forces with the help of equation (8) as

\[
\{F_i\} = \{K_{xx}\} \{U_i\} + \{K_{yy}\} \{U_i\}
\]

It becomes as

\[
K_{15}U_5 + K_{16}U_6 + \cdots + K_{18}U_8 = F_i, \ i = 1, 2, 3, 4
\]

For i=1:

\[
F_1 = K_{15}U_5 + K_{16}U_6 + \cdots + K_{18}U_8
\]

Similarly for For i=2, 3, 4

\[
F_2 = K_{25}U_5 + K_{26}U_6 + \cdots + K_{28}U_8
\]

Therefore, the system of reaction forces is
\[
\begin{bmatrix}
F_1 \\
F_2 \\
F_3
\end{bmatrix} =
\begin{bmatrix}
-12000 \\
-4000 \\
6000 \\
0
\end{bmatrix} \text{lb}
\]

Now we shall find element displacements
For Element - 1:
\[
u_1^{(1)} = U_1 \cos \theta_1 + U_2 \sin \theta_1 = 0 + 0 = 0
\]
\[
u_2^{(1)} = U_3 \cos \theta_1 + U_4 \sin \theta_1 = (0.02133) \cos 0^\circ + (0.04085) \sin 0^\circ
\]
\[
= 0.02133
\]
Similarly for Element i=2, 3, 4, 5, 6, 7, 8:
\[
u_1^{(2)} = 0, \quad u_1^{(3)} = 0
\]
\[
u_2^{(2)} = 0.021348, \quad u_2^{(3)} = -0.01600
\]
\[
u_1^{(4)} = 0.04085, \quad u_2^{(5)} = 0.02133
\]
\[
u_2^{(4)} = 0.04619, \quad u_2^{(5)} = 0.04267
\]
\[
u_1^{(6)} = 0.075993, \quad u_2^{(7)} = -0.01600
\]
\[
u_2^{(6)} = 0.043975, \quad u_2^{(7)} = -0.00533
\]
\[
u_1^{(8)} = 0.15014
\]
\[
u_2^{(8)} = 0.16614
\]

Now we shall find axial strain for each element
\[
\varepsilon^{(e)} = \frac{u_2^{(e)} - u_1^{(e)}}{L^{(e)}}
\]
For Element i=1, 2, 3, 4, 5, 6, 7, 8:
\[
\varepsilon^{(1)} = 5.33 \times 10^{-4}, \quad \varepsilon^{(2)} = 3.77 \times 10^{-4}
\]
\[
\varepsilon^{(3)} = -4.00 \times 10^{-4}, \quad \varepsilon^{(4)} = 1.34 \times 10^{-4}
\]
\[
\varepsilon^{(5)} = 5.34 \times 10^{-4}, \quad \varepsilon^{(6)} = -5.66 \times 10^{-4}
\]
\[
\varepsilon^{(7)} = 2.67 \times 10^{-4}, \quad \varepsilon^{(8)} = 4.00 \times 10^{-4}
\]

Now we will find corresponding axial stress for each elements by using equation
\[
\sigma^{(e)} = E\varepsilon^{(e)}
\]
For Element i=1, 2, 3, 4, 5, 6, 7, 8:
\[
\sigma^{(1)} = 5.33 \times 10^3, \quad \sigma^{(2)} = 3.77 \times 10^3
\]
\[
\sigma^{(3)} = -4.00 \times 10^3, \quad \sigma^{(4)} = 1.34 \times 10^3
\]
\[
\sigma^{(5)} = 5.34 \times 10^3, \quad \sigma^{(6)} = -5.66 \times 10^3
\]
\[
\sigma^{(7)} = 2.67 \times 10^3, \quad \sigma^{(8)} = 4.00 \times 10^3
\]

<table>
<thead>
<tr>
<th>Element</th>
<th>Strain</th>
<th>Stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5.33 \times 10^{-4}</td>
<td>5330</td>
</tr>
<tr>
<td>2</td>
<td>3.77 \times 10^{-4}</td>
<td>3770</td>
</tr>
<tr>
<td>3</td>
<td>-4.00 \times 10^{-4}</td>
<td>-4000</td>
</tr>
<tr>
<td>4</td>
<td>1.34 \times 10^{-4}</td>
<td>1340</td>
</tr>
<tr>
<td>5</td>
<td>5.34 \times 10^{-4}</td>
<td>5340</td>
</tr>
<tr>
<td>6</td>
<td>-5.66 \times 10^{-4}</td>
<td>-5660</td>
</tr>
<tr>
<td>7</td>
<td>2.67 \times 10^{-4}</td>
<td>2670</td>
</tr>
<tr>
<td>8</td>
<td>4.00 \times 10^{-4}</td>
<td>4000</td>
</tr>
</tbody>
</table>

**Figure:** 3 Graph for eight elements values of strain and stress.

4. CONCLUSION
Two linear mechanical elements, the idealized elastic spring and an elastic tension compression member (bar) have been used to introduce the basic concepts involved in formulating the equations governing a finite element. The element equations are obtained by both a straightforward equilibrium approach and a strain energy method. The principle of minimum potential is also introduced. The one-dimensional bar element can be used to demonstrate the finite element model assembly procedures in the context of some simple two and three dimensional structures.

This research develops the complete procedure for performing a finite element analysis of a structure and illustrates it by several examples. Although only the simple axial element has been used, the procedure described is common to the finite element method for all element and analysis. The direct stiffness method is by far the most straightforward technique for assembling the system matrices required for finite element analysis and is also very amenable to digital computer programming techniques.

**REFERENCES**


