FINITE ELEMENT METHOD

Homework 2

KIRAN SAGAR KOLLEPARA
Computational Mechanics

The strong form of the equation is:

$$\nabla \cdot \boldsymbol{\sigma} + \boldsymbol{b} = 0$$

And the boundary conditions can be stated as:

$$u(x, y=0) = 0 \ , v(x, y=0) = 0$$
$$u(x=0, y) = 0$$

The first boundary condition corresponds to the fixed boundary on the bottom, and the second boundary condition corresponds to the symmetry of the boundary condition.

Actually, another boundary conditions of stresses being continuous as well as symmetric is applicable to the plane of symmetry as follows:

$$\frac{\partial u}{\partial y} = 0 \ \& \ \frac{\partial v}{\partial x} = 0$$

However, this condition need NOT be applied, since the weak form of the governing equation is being used. Furthermore, these conditions may not be satisfied by the FE solution.

The nodes of each element are assigned local number as: # 1 at the right angled vertex and # 2 and # 3 for the other nodes in a manner that the arrangement of the nodes is clockwise in ascending order, as shown in Figure 1.

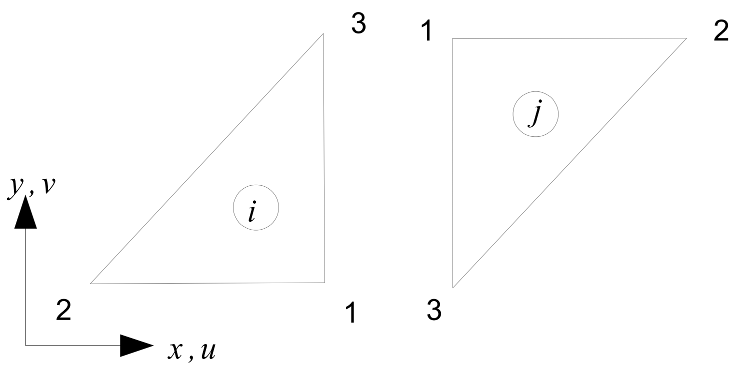

Figure 1: Local numbering of element

Using this numbering scheme, the connectivity matrix can be defined as follows:

$$\boldsymbol{T} = \begin{bmatrix} 2 & 1 & 4 \\ 4 & 5 & 2 \\ 3 & 2 & 5 \\ 5 & 4 & 6 \end{bmatrix}$$

along with the nodal coordinates as:

$$\boldsymbol{X} = \begin{bmatrix} -3 & -1.5 & 0 & -1.5 & 0 & 0 \\ 0 & 0 & 0 & 1.5 & 1.5 & 3 \end{bmatrix}^T$$

The stiffness matrix would be of a single element would be of size 6 x 6, with each node having two degrees of freedom and a 2 x 2 matrix associated with it.

$$\boldsymbol{K}^{(e)} = \begin{bmatrix} \boldsymbol{K}_{11}^{(e)} & \boldsymbol{K}_{12}^{(e)} & \boldsymbol{K}_{13}^{(e)} \\ \boldsymbol{K}_{21}^{(e)} & \boldsymbol{K}_{22}^{(e)} & \boldsymbol{K}_{23}^{(e)} \\ \boldsymbol{K}_{31}^{(e)} & \boldsymbol{K}_{32}^{(e)} & \boldsymbol{K}_{33}^{(e)} \end{bmatrix}, \text{ where } \boldsymbol{K}_{ij}^{(e)} = \begin{bmatrix} K_{ij,uu}^{(e)} & K_{ij,uv}^{(e)} \\ K_{ij,vu}^{(e)} & K_{ij,,vv}^{(e)} \end{bmatrix}.$$

Using the element connectivity matrix, we can obtain the element *e* contribution to the assembly matrix using:

$$\boldsymbol{K}^{(e)}_{global} = \boldsymbol{R}^{(e)} T\, \boldsymbol{K}^{(e)} \boldsymbol{R}^{(e)}$$

where, $\boldsymbol{R}^{(e)} = \begin{bmatrix} \mathbf{0} & \mathbf{0} & .... & .... & .... & \mathbf{0} & \boldsymbol{I} & \mathbf{0} & .... & .... \\ \mathbf{0} & \mathbf{0} & .... & \mathbf{0} & \boldsymbol{I} & \mathbf{0} & .... & .... & .... & .... \\ \mathbf{0} & \mathbf{0} & .... & .... & .... & .... & \mathbf{0} & \boldsymbol{I} & \mathbf{0} & .... \end{bmatrix}$, $\mathbf{0} = \begin{bmatrix} 0 & 0 \\ 0 & 0 \end{bmatrix}$, $\boldsymbol{I} = \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}$

The matrix $\boldsymbol{R}$ is sized *6* x *2n,* with *n* as the total number of nodes. In each pair of rows, we have an *2* x *2* identity matrix at the position of each node i.e. T_{11}, T_{12}, T_{13}, and otherwise zero.

So, the complete assembly matrix would be:

$$\boldsymbol{K}_{global} = \sum_e \boldsymbol{R}^{(e)T} \boldsymbol{K}^{(e)} \boldsymbol{R}^{(e)}$$

Using this assembly procedure, the following assembly procedure can be obtained:

$$\boldsymbol{K}_{global} = \begin{bmatrix} \boldsymbol{K}^{(1)}_{22} & \boldsymbol{K}^{(1)}_{21} & 0 & \boldsymbol{K}^{(1)}_{23} & 0 & 0 \\ \boldsymbol{K}^{(1)}_{12} & \boldsymbol{K}^{(1)}_{11} + \boldsymbol{K}^{(3)}_{22} + \boldsymbol{K}^{(2)}_{33} & \boldsymbol{K}^{(3)}_{21} & \boldsymbol{K}^{(1)}_{13} + \boldsymbol{K}^{(2)}_{31} & \boldsymbol{K}^{(3)}_{23} + \boldsymbol{K}^{(2)}_{32} & 0 \\ 0 & \boldsymbol{K}^{(3)}_{12} & \boldsymbol{K}^{(3)}_{11} & 0 & \boldsymbol{K}^{(3)}_{13} & 0 \\ \boldsymbol{K}^{(1)}_{32} & \boldsymbol{K}^{(2)}_{13} + \boldsymbol{K}^{(1)}_{31} & 0 & \boldsymbol{K}^{(2)}_{11} + \boldsymbol{K}^{(4)}_{22} + \boldsymbol{K}^{(1)}_{33} & \boldsymbol{K}^{(2)}_{12} + \boldsymbol{K}^{(4)}_{21} & \boldsymbol{K}^{(4)}_{23} \\ 0 & \boldsymbol{K}^{(2)}_{23} + \boldsymbol{K}^{(3)}_{32} & \boldsymbol{K}^{(3)}_{31} & \boldsymbol{K}^{(4)}_{12} + \boldsymbol{K}^{(2)}_{21} & \boldsymbol{K}^{(4)}_{11} + \boldsymbol{K}^{(2)}_{22} + \boldsymbol{K}^{(3)}_{33} & \boldsymbol{K}^{(4)}_{13} \\ 0 & 0 & 0 & \boldsymbol{K}^{(4)}_{32} & \boldsymbol{K}^{(4)}_{31} & \boldsymbol{K}^{(4)}_{33} \end{bmatrix}$$

The same matrix $\boldsymbol{R}$ can be used to generate the body force vector as follows:

$$\boldsymbol{F}_b = \sum_e \boldsymbol{R}^{(e)T} \boldsymbol{b}^{(e)}$$

where, $\boldsymbol{b}$ is the body force vector for each element.

The shape functions for the isosceles right triangular element, shown in Figure 2, are:

$$N_1 = 1 + \frac{2}{3}(x - y), \quad N_2 = \frac{-2}{3} x, \quad N_3 = \frac{-2}{3} y$$

with the 1st node as the right angle, and rest nodes arranged clockwise.

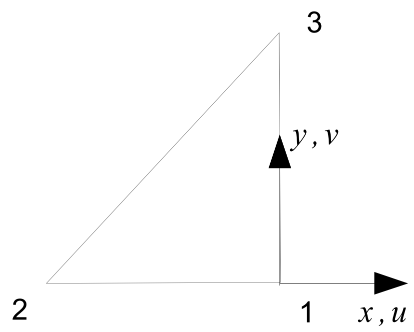

Figure 2: Local co-ordinate system

Using the theory for plane stress, we have:

$$\boldsymbol{K} = \iint_{A_e} \boldsymbol{B}^T \boldsymbol{D} \boldsymbol{B}\, t\, dA_e$$

where, $\boldsymbol{B} = [\boldsymbol{B}_1 \quad \boldsymbol{B}_2 \quad \boldsymbol{B}_3]$,

$$\boldsymbol{B}_i = \begin{bmatrix} \frac{dN_i}{dx} & 0 \\ 0 & \frac{dN_i}{dy} \\ \frac{dN_i}{dy} & \frac{dN_i}{dx} \end{bmatrix} \text{ and } \boldsymbol{D} = E \begin{bmatrix} \frac{-1}{(nu^2-1)} & \frac{-nu}{(nu^2-1)} & 0 \\ \frac{-nu}{(nu^2-1)} & \frac{-1}{(nu^2-1)} & 0 \\ 0 & 0 & \frac{1}{(2(nu+1))} \end{bmatrix}$$

On evaluating this integral using the parameters provided, a 6x6 matrix is obtained:

$$
\boldsymbol{K} = 1 \text{ x } 10^9 \begin{bmatrix} 7.2917 & -3.1250 & -5.2083 & 2.0833 & -2.0833 & 1.0417 \\ -3.1250 & 7.2917 & 1.0417 & -2.0833 & 2.0833 & -5.2083 \\ -5.2083 & 1.0417 & 5.2083 & 0 & 0 & -1.0417 \\ 2.0833 & -2.0833 & 0 & 2.0833 & -2.0833 & 0 \\ -2.0833 & 2.0833 & 0 & -2.0833 & 2.0833 & 0 \\ 1.0417 & -5.2083 & -1.0417 & 0 & 0 & 5.2083 \end{bmatrix}
$$

Similarly, the force matrix can be obtained:

$$
\boldsymbol{b} = \iint_{A_e} \boldsymbol{N}^T \boldsymbol{b} \, t \, dA_e
$$

$$
\Rightarrow \boldsymbol{b} = -\frac{3}{8} \begin{bmatrix} 0 & \rho g & 0 & \rho g & 0 & \rho g \end{bmatrix}^T
$$

Element #2 can be obtained by rotating the reference element by 180^0. Hence, x-y axis and u-v directions are also rotated by 180^0. Suppose x', y', u' and v' are the coordinates of the reference elements. So, the vectors in global coordinates are related to reference coordinates as:

$$
x = -x', \; y = -y'
$$

$$
u = -u', \; v = -v'
$$

$$
F_x = -F_{x'}, \; F_y = -F_{y'}
$$

Hence, the relationship between the stiffness matrix in both the coordinate systems can be established as follows:

$$
\boldsymbol{F} = -\boldsymbol{F}'
$$

$$
\Rightarrow \boldsymbol{K} \boldsymbol{u} = -\boldsymbol{K}' \boldsymbol{u}'
$$

Since, $\boldsymbol{u} = -\boldsymbol{u}'$

$$
\Rightarrow \boldsymbol{K} = \boldsymbol{K}'
$$

Hence, the stiffness matrix of the rotated element is same as other elements. Based on these results we can construct the global stiffness matrix and force matrix as follows:

$\boldsymbol{K}_{global} = 10^{10}$ x

$$
\begin{bmatrix}
0.5208 & 0 & -0.5208 & 0.1042 & 0 & 0 & 0 & -0.1042 & 0 & 0 & 0 & 0 \\
0 & 0.2083 & 0.2083 & -0.2083 & 0 & 0 & -0.2083 & 0 & 0 & 0 & 0 & 0 \\
-0.5208 & 0.2083 & 1.4583 & -0.3125 & -0.5208 & 0.1042 & -0.4167 & 0.3125 & 0 & -0.3125 & 0 & 0 \\
0.1042 & -0.2083 & -0.3125 & 1.4583 & 0.2083 & -0.2083 & 0.3125 & -1.0417 & -0.3125 & 0 & 0 & 0 \\
0 & 0 & -0.5208 & 0.2083 & 0.7292 & -0.3125 & 0 & 0 & -0.2083 & 0.1042 & 0 & 0 \\
0 & 0 & 0.1042 & -0.2083 & -0.3125 & 0.7292 & 0 & 0 & 0.2083 & -0.5208 & 0 & 0 \\
0 & -0.2083 & -0.4167 & 0.3125 & 0 & 0 & 1.4583 & -0.3125 & -1.0417 & 0.3125 & 0 & -0.1042 \\
-0.1042 & 0 & 0.3125 & -1.0417 & 0 & 0 & -0.3125 & 1.4583 & 0.3125 & -0.4167 & -0.2083 & 0 \\
0 & 0 & 0 & -0.3125 & -0.2083 & 0.2083 & -1.0417 & 0.3125 & 1.4583 & -0.3125 & -0.2083 & 0.1042 \\
0 & 0 & -0.3125 & 0 & 0.1042 & -0.5208 & 0.3125 & -0.4167 & -0.3125 & 1.4583 & 0.2083 & -0.5208 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & -0.2083 & -0.2083 & 0.2083 & 0.2083 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & -0.1042 & 0 & 0.1042 & -0.5208 & 0 & 0.5208
\end{bmatrix}
$$

$$
\boldsymbol{F}_{bodyforce} = \begin{bmatrix} 0 & -375 & 0 & -1125 & 0 & -375 & 0 & -1125 & 0 & -1125 & 0 & -375 \end{bmatrix}^T
$$

$$
\boldsymbol{F}_{total} = \boldsymbol{F}_{bodyforces} + \begin{bmatrix} R_{1,x} & R_{1,y} & R_{2,x} & R_{2,y} & R_{3,x} & R_{3,y} & 0 & 0 & R_{5,x} & 0 & R_{6,x} & F_{6,y} \end{bmatrix}^T
$$

By applying boundary conditions, we eliminate the variables, both 'u' & 'v' of node 1,2 and 3, as well as 'u' of node 5 & 6. Also, the vertical displacement at node 6 is given, which eliminates the 'v' of node 6. The remaining variables are: $u_{4,} v_{4,} v_5$.

Thus, the number of degrees of freedom $= 3$

The force vector has to be modified based on the prescribed value of nodal displacements.

$$\boldsymbol{F}_{total,with\ BC's} = \boldsymbol{F}_{total} - \sum_j \boldsymbol{K}_{j^{th}\ column} u_j$$

where u_j is the jth node with a prescribed boundary value or given displacement.

After the force vector is modified to account for known boundary displacements, we can eliminate all the jth row and column from the stiffness matrix and jth row from the force vectors. The resultant system obtained is:

$$\begin{bmatrix} 1.4583 & -0.3125 & 0.3125 \\ -0.3125 & 1.4583 & -0.4167 \\ 0.3125 & -0.4167 & 1.4583 \end{bmatrix} \text{x } 10^{10} \begin{bmatrix} u_4 \\ v_4 \\ v_5 \end{bmatrix} = \begin{bmatrix} -1.0416667 \\ -0.0001125 \\ 0 \\ -5.2084458 \end{bmatrix} \text{x } 10^{7}$$

and the resulting solution is obtained by inverting the stiffness matrix:

$$\boldsymbol{u}_h = \begin{bmatrix} u_4 \\ v_4 \\ v_5 \end{bmatrix} = \begin{bmatrix} -0.128205 \\ -1.132586 \\ -3.867629 \end{bmatrix} \text{x } 10^{-3} \approx \begin{bmatrix} -0.1282\ mm \\ -1.1326\ mm \\ -3.8676\ mm \end{bmatrix}$$

* * *END * * *
