UNIVERSITAT POLITÈCNICA DE CATALUNYA BARCELONATECH

HOMEWORK ASSIGNMENT 2

Waleed Ahmad Mirza

Course: Finite Element Methods

Contents

1-Governing Equation (Strong form)

$div\sigma + \underline{b} = \rho\underline{a}$ in Ω

$\bar{t} - \sigma n = 0$ on $\partial\sigma\Omega$

$\underline{u} = \bar{u}$ on $\partial u\Omega$

For the given case study in tensoral form the governing equation can be written as,

In tensor form of the above equation can be written as,

$$\begin{bmatrix} \partial/\partial x & 0 & \partial/\partial y \\ 0 & \partial/\partial y & \partial/\partial x \end{bmatrix} \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} + \begin{bmatrix} b_x \\ b_y \end{bmatrix} = \begin{bmatrix} \rho a_x \\ \rho a_y \end{bmatrix}$$

2- Driving Weak Form using Virtual work

The variational problem, known as the virtual work principle, may be written as: Find an admissible displacement field $u(x) \in U$ such that,

$$\delta W(u, \delta u) = DW(u).\delta u$$

$$= \int_{\Omega} E \cdot \delta\underline{u} d\Omega + \int_{\partial\sigma\Omega} T \cdot \delta\underline{u} d\Gamma$$

$$= \int_{\Omega} (div\sigma + \rho\underline{b}) \cdot \delta\underline{u} d\Omega + \int_{\partial\sigma\Omega} (\bar{t} - \sigma n) \cdot \delta\underline{u} d\Gamma = o \qquad \forall \delta u \in U_0 \qquad (1)$$

Where $\delta W(\underline{u}, \delta\underline{u})$ is the Gateaux derivative of a functional and can be viewed as the total virtual work. Note that in equation 1, the system is considered quasi-static. Hence acceleration term is neglected. Now using divergence theorem.

$$\int_{\Omega} div\sigma \cdot \delta\underline{u} d\Omega = \int_{\Omega} div(\delta\underline{u} \cdot \sigma) d\Omega - \int_{\Omega} (\sigma : grad\delta\underline{u}) d\Omega$$

$$\int_{\Omega} div\sigma \cdot \delta\underline{u} d\Omega = \int_{\partial\Omega} \delta\underline{u} \cdot (\sigma n) d\mathrm{T} - \int_{\Omega} (\sigma : grad\delta\underline{u}) d\Omega$$

$$\int_{\Omega} div\sigma \cdot \delta\underline{u} d\Omega = \int_{\partial\sigma\Omega} \delta\underline{u} \cdot (\sigma n) d\mathrm{T} - \int_{\Omega} \sigma : grad\delta\underline{u} d\Omega$$

Substituting the term in equation 1, we get following:

$$\delta W(u, \delta u) = DW(u).\delta u) = 0$$

$$-\int_{\Omega} \sigma : grad\delta\underline{u} d\Omega + \int_{\Omega} \rho\underline{b} \cdot \delta\underline{u} d\Omega + \int_{\partial\sigma\Omega} \bar{t} \cdot \delta\underline{u} d\mathrm{T}' = 0$$

$$\delta W_{\text{int}} = \int_{\Omega} \sigma : \delta \varepsilon d\Omega$$

$$\delta W_{out} = \int_{\Omega} \rho \underline{b} \cdot \delta \underline{u} d\Omega + \int_{\partial\sigma\Omega} \bar{t} \cdot \delta \underline{u} d\mathrm{T}$$

Hence

$$\delta W(\underline{u}, \delta \underline{u}) = DW(\underline{u}).\delta(\underline{u}) = \delta W_{out} - \delta W_{\text{int}} = 0 \tag{2}$$

For two dimensional element in component form the above equation can be written as,

$$\delta W_{\text{int}} = \int \sigma_{xx} \delta\varepsilon_{xx} + \sigma_{yy} .\delta\varepsilon_{yy} + \sigma_{xy} \delta\varepsilon_{xy}) d\Omega$$
$$\delta W_{out} = \iint_A (\delta u b_x + \delta v b_y) t.dA + \oint_l (\delta u \bar{t}_x + \delta v \bar{t}_y) t ds + \sum_i (\delta u_i U_i + \delta v_i V_i)$$

Where,

$$\delta \underline{\varepsilon} = \left[\delta\varepsilon_x, \delta\varepsilon_y, \delta\gamma_{xy}\right]^T$$
$$\delta \underline{u} = \left[\delta u, \delta v\right]^T$$
$$\underline{b} = \left[b_x, b_y\right]^T$$
$$\bar{t} = \left[\bar{t}_x, \bar{t}_y\right]^T$$
$$\delta \underline{u}_i = \left[\delta u_i, \delta v_i\right]^T$$
$$\underline{q_i} = \left[U_i, V_i\right]^T$$

Putting the above two equations in equation (2).

$$\iint_A (\delta\varepsilon_{xx}\sigma_{xx} + \delta\varepsilon_{yy}\sigma_{yy} + \delta\gamma_{xy}\tau_{xy}) t dA = \iint_A (\delta u b_x + \delta v b_y) t.dA + \oint_l (\delta u \bar{t}_x + \delta v \bar{t}_y) t ds + \sum_i (\delta u_i U_i + \delta v_i V_i) \tag{3}$$

Moving back to tensor formulation. Now here onwards, equations are particularized for individual discretised elements and Voigt notations are used for stress and strain, that is why they will be presented with an underscore symbol.

$$\iint_{A^{(e)}} \delta \underline{\varepsilon}^T : \underline{\sigma} t dA = \iint_{A^{(e)}} \delta \underline{u}^T . \underline{b} t dA + \oint_{l^{(e)}} \delta \underline{u}^T . \bar{t} t ds + \sum_i \delta \underline{u}_i^T \underline{q}_i \tag{4}$$

Next we interpolate the virtual displacements in terms of the nodal values and obtain the following equation in terms of shape function vector N.

$$\left[\delta \underline{u}^{(e)}\right]^T \left[\iint_{A^{(e)}} B^T \underline{\sigma} t dA - \iint_{A^{(e)}} \underline{N}^T \underline{b} t dA - \oint_{l^{(e)}} \underline{N}^T \bar{t} t dS \right] = \left[\delta \underline{u}^{(e)}\right]^T \underline{q}^{(e)} \tag{5}$$

Since the virtual displacements are arbitrary it is finally deduced that

$$\iint_{A^{(e)}} B^T \underline{\sigma} t dA - \iint_{A^{(e)}} \underline{N}^T \underline{b} t dA - \oint_{l^{(e)}} \underline{N}^T \bar{t} t ds = \underline{q}^{(e)} \tag{6}$$

Substituting the stresses in terms of the nodal displacements from Equation (6) gives

$$\iint_{A^{(e)}} B^T D B \underline{u}^{(e)} t dA - \iint_{A^{(e)}} \underline{N}^T \underline{b} t dA - \oint_{l^{(e)}} \underline{N}^T \bar{t} t ds = \underline{q}^{(e)} \tag{7}$$

And

$$\left[\iint_{A^{(e)}} B^T D B t dA \right] \underline{u}^{(e)\prime} - \iint_{A^{(e)}} \underline{N}^T \underline{b} t dA - \oint_{l^{(e)}} \underline{N}^T \bar{t} t ds = \underline{q}^{(e)} \tag{8}$$

Or

$$K^{(e)} \underline{u}^{(e)} - \underline{f}^{(e)} = \underline{q}^{(e)} \tag{9}$$

Where

$$K^{(e)} = \iint_{A^{(e)}} B^T D B t dA \tag{10}$$

Is the element of stiffness matrix, and

$$\underline{f}^{(e)} = \underline{f}_b^{(e)} + \underline{f}_t^{(e)} \tag{11}$$

Is the equivalent nodal force vector for the element where

$$\underline{f}_b^{(e)} = \iint_{A^{(e)}} \underline{N}^T \underline{b} t dA \tag{11}$$

$$\underline{f}_t^{(e)} = \oint_{l^{(e)}} \underline{N}^T \bar{t} t ds \tag{12}$$

The global equilibrium equations for the whole mesh are obtained by establishing that the nodes are in equilibrium i.e. the sum of all the equilibrating nodal forces at a node balance the external point loads and

$$\sum_e \underline{q}_i^{(e)} = \underline{p}_j \tag{13}$$

Where p_j represents the vector of external point loads acting at node j and the sum refers to all elements sharing the node. The global equilibrium equation is written in matrix form as

$$K\underline{U} = \underline{F} \tag{16}$$

Where K and f are the global stiffness matrix and the equivalent nodal force vector for the whole mesh. Weak formulation of the given system is summed up in figure (1) as Tonti Diagram.

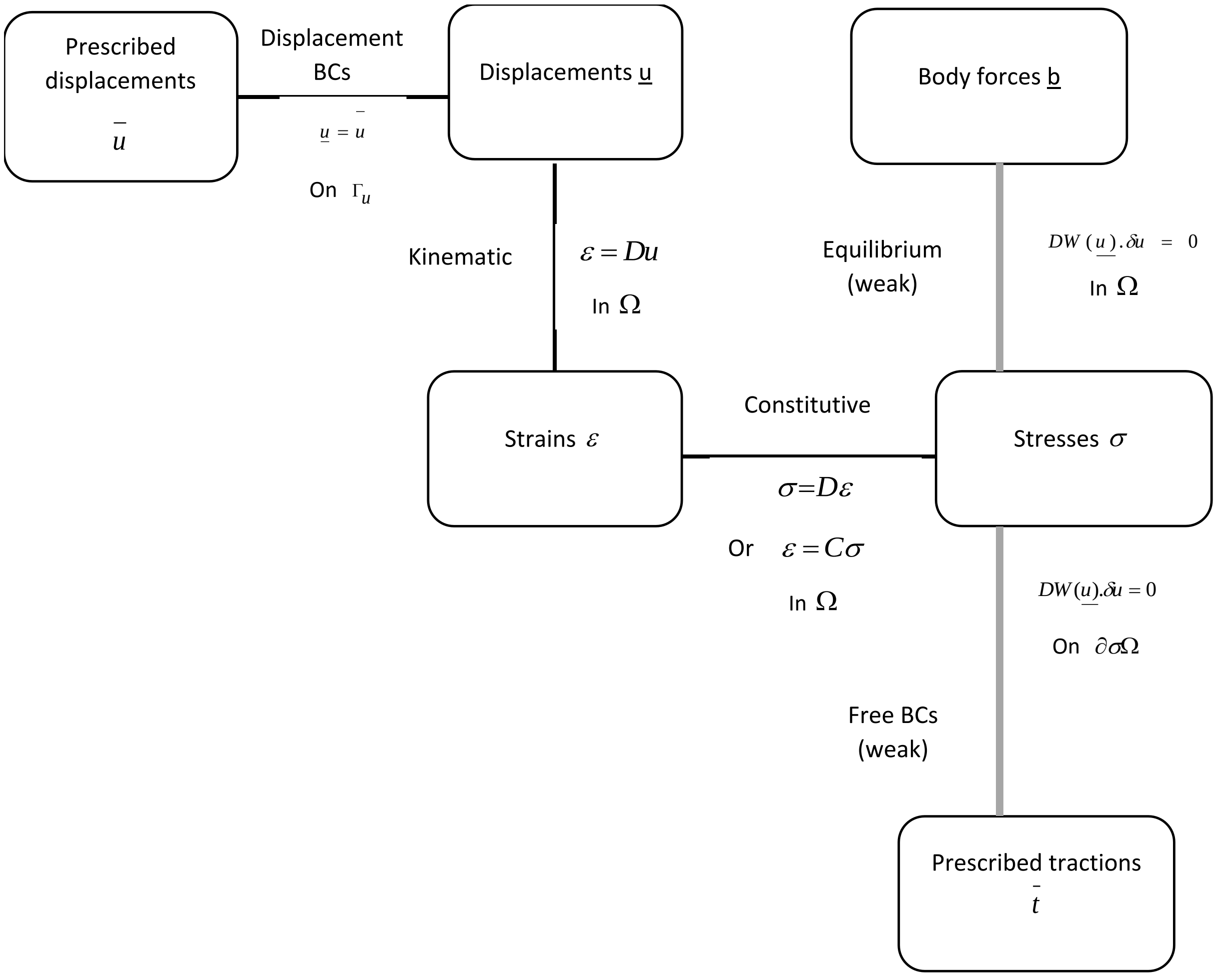

Figure 1:Tonti Diagram

3-Planar Triangular element

Three node triangular element is the simplest of all planar elements. The shape functions are simply the triangular coordinates. That is, $N_i^e = \zeta_i$ for i=1, 2,3. For the plane stress problem we select the linear interpolation for the displacement components u_x and u_y at an arbitrary point $P = (\zeta_1, \zeta_2, \zeta_3)$.

$$u = u_1\zeta_1 + u_2\zeta_2 + u_3\zeta_3, v = v_1\zeta_1 + v_2\zeta_2 + v_3\zeta_3 \quad (17)$$

$$\begin{bmatrix} u \\ v \end{bmatrix} = \begin{bmatrix} \zeta_1 & 0 & \zeta_2 & 0 & \zeta_3 & 0 \\ 0 & \zeta_1 & 0 & \zeta_2 & 0 & \zeta_3 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix} = N\underline{u}^e, \quad (18)$$

Where N is the matrix of shape functions. The strains within the elements are obtained by differentiating the shape functions with respect to x and y. Therefore, we get

$$\underline{\varepsilon} = B\underline{u}^e = \frac{1}{2A}\begin{bmatrix} y_{23} & 0 & y_{31} & 0 & y_{12} & 0 \\ 0 & x_{32} & 0 & x_{13} & 0 & x_{21} \\ x_{32} & y_{23} & x_{13} & y_{31} & x_{21} & y_{12} \end{bmatrix}\begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix}, \tag{19}$$

in which B is the strain-displacement matrix and $y_{ij} = y_i - y_j$, $x_{ij} = x_i - x_j$. Note that the strains are constant over the element. The stress field σ is related to the strain field by the elastic constitutive equation which is as follows:

$$\underline{\sigma} = \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = \begin{bmatrix} D_{11} & D_{12} & D_{13} \\ D_{12} & D_{22} & D_{23} \\ D_{13} & D_{23} & D_{33} \end{bmatrix}\begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ 2\varepsilon_{xy} \end{bmatrix} = D\underline{\varepsilon}, \tag{20}$$

Where D_{ij} are the plane stress elastic moduli. The constitutive matrix D will be assumed to be constant over the element. Because the strains are constant, so are the stress.

The element stiffness matrix is given by the following formula:

$$\mathrm{K}^e = \int_{\Omega^e} t\mathrm{B}^T D\mathrm{B}\,d\Omega, \tag{21}$$

Where Ω^e is the triangle domain, and t the plate thickness that appears in the plane stress problem. Since B and D are constant, they can be taken out of the integral:

$$\mathrm{K}^e = \mathrm{B}^T\mathbf{D}\mathrm{B}\int_{\Omega^e} t\,d\Omega \tag{22}$$

If t is uniform over the element the remaining integral in equation above is simply tA, and we obtain the closed form

$$\mathrm{K}^e = At\mathrm{B}^T\mathrm{D}\mathrm{B} = \frac{t}{4A}\begin{bmatrix} y_{23} & 0 & x_{32} \\ 0 & x_{32} & y_{23} \\ y_{31} & 0 & x_{13} \\ 0 & x_{13} & y_{31} \\ y_{12} & 0 & x_{21} \\ 0 & x_{21} & y_{12} \end{bmatrix}\begin{bmatrix} D_{11} & D_{12} & D_{13} \\ D_{12} & D_{22} & D_{23} \\ D_{13} & D_{23} & D_{33} \end{bmatrix}\begin{bmatrix} y_{23} & 0 & y_{31} & 0 & y_{12} & 0 \\ 0 & x_{32} & 0 & x_{13} & 0 & x_{21} \\ x_{32} & y_{23} & x_{13} & y_{31} & x_{21} & y_{12} \end{bmatrix}$$

(23)

For simplicity we consider here only body forces defined by the vector field:

$$\underline{\mathbf{b}} = \begin{bmatrix} b_x \\ b_y \end{bmatrix} \tag{24}$$

Which is specified per unit of volume. The consistent nodal force vector $\mathbf{f}^e$ is given by the following formula:

$$\underline{\mathrm{f}_b}^e = \int_{\Omega^e} t\mathrm{N}^T \mathrm{b} d\Omega = \int_{\Omega^e} t \begin{bmatrix} \zeta_1 & 0 \\ 0 & \zeta_1 \\ \zeta_2 & 0 \\ 0 & \zeta_2 \\ \zeta_3 & 0 \\ 0 & \zeta_3 \end{bmatrix} \mathrm{b} d\Omega \tag{25}$$

The simplest case in when the body force components as well as thickness t are constant over the element. Then we need the integrals

$$\int_{\Omega^e} \zeta_1 d\Omega = \int_{\Omega^e} \zeta_2 d\Omega = \int_{\Omega^e} \zeta_3 d\Omega = \frac{1}{3} A \tag{26}$$

Which replaced into equation no.8 gives the following:

$$\underline{f_b}^e = \frac{At}{3} \begin{bmatrix} b_x & b_y & b_x & b_y & b_x & b_y \end{bmatrix}^T \tag{27}$$

Formulation of traction force vector is as follow.

Shape function of a node not belonging to the loaded boundary has a zero value. This, if the element side 1-2 is loaded with uniformly distributed tractions t_x and t_y, vector $\underline{\mathrm{f}_t^{(e)}}$ becomes

$$\underline{\mathrm{f}_t^{(e)}} = \frac{(l_{12}t)^{(e)}}{2} \begin{Bmatrix} \overline{t_x} \\ \overline{t_y} \\ \overline{t_x} \\ \overline{t_y} \\ 0 \\ 0 \end{Bmatrix} \tag{28}$$

Where $l_{12}^{(e)}$ is the side length. Equation no.1 shows that the traction force acting along the element side is distributed into equal parts between the two side nodes. The expressions of $\mathbf{f}_t^{(e)}$ for loaded sides 1-3 and 2-3 are as follows:

$$\underline{\mathrm{f}_t^{(e)}} = \frac{(l_{13}t)^{(e)}}{2} \begin{Bmatrix} \overline{t_x} \\ \overline{t_y} \\ 0 \\ 0 \\ \overline{t_x} \\ \overline{t_y} \end{Bmatrix}; \underline{\mathrm{f}_t^{(e)}} = \frac{(l_{23}t)^{(e)}}{2} \begin{Bmatrix} 0 \\ 0 \\ \overline{t_x} \\ \overline{t_y} \\ \overline{t_x} \\ \overline{t_y} \end{Bmatrix} \qquad (30), (31)$$

4-Case study

In the given case study described in the figure (2) the structure at hand is symmetric about y = 0 Hence only left half of the structure is considered for analysis. As given the structure is divided in 4 elements with fixed Drichelet boundary condition applied on node 1, 2, 3 and restricted Drichelet boundary condition of δ= 0.01 m applied on node 6. For the material matrix D, the given input data is as follow,

E = 10GPa , ν = 0.2 , ρg = 10^3 N/m2

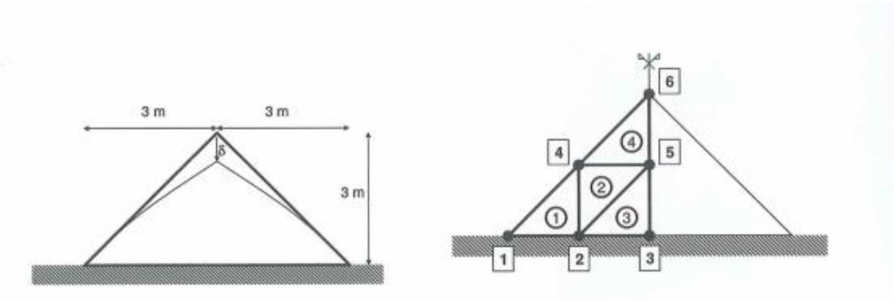

Figure 2: Mesh and geometric description

Solution

The given case study in solved by using the formulation given in the previous sections.

4.1-Connectivity Matrix

The first step is to assign connectivity matrix in order to outline the relationship between the local and global nod numbering. As given in the problem description, the node at right angle of the triangle is given local node numbering 1 and successive numbers are assigned in the counter clockwise direction. The advantage of consistent node numbering is that for all dimensionally similar elements the stiffness matrix formulation is same. The connectivity matrix displaying global node numbering goes as follow.

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

ith row of connectivity matrix represents local node numbering of the ith element.

4.2-Prescribed boundary conditions

Next step is to prescribe boundary condition. As given in the problem description, fixed displacement boundary conditions are applied on node 1, 2, 3 , whereas vertical displacement is applied δ = v = 0.01 on node 6. Since symmetry of the structure about y = 0 is utilized hence boundary condition of $u_{5x} = u_{6x} = 0$ is applied on node 5 and 6.

4.3-Calculating Global Stiffness Matrix

Now in order to find the stiffness matrix $K^{(e)}$ and strain displacement matrix $B^{(e)}$ of each element we will use the formulas described in previous sections. As a result the given strain displacement matrix are evaluated by using equation (19) are as follow.

$$B^{(1)} = B^{(2)} = B^{(3)} = B^{(4)} = \begin{bmatrix} 1.500 & 0 & 0 & 0 & -1.500 & 0 \\ 0 & -1.500 & 0 & 1.500 & 0 & 0 \\ -1.500 & 1.500 & 1.500 & 0 & 0 & -1.500 \end{bmatrix}$$

Material and local stiffness matrix (order 6x6) are obtained by the using equation (23)

$$K^{(1)} = K^{(2)} = K^{(3)} = K^{(4)} = 10E^{9} \times \begin{bmatrix} 7.2917 & -3.1250 & -2.0833 & 1.0417 & -5.2083 & 2.0833 \\ -3.1250 & 7.2917 & 2.0833 & -5.2083 & 1.0417 & -2.0833 \\ -2.0833 & 2.0833 & 2.0833 & 0 & 0 & -2.0833 \\ 1.0417 & -5.2083 & 0 & 5.2083 & -1.0417 & 0 \\ -5.2083 & 1.0417 & 0 & -1.0417 & 5.2083 & 0 \\ 2.0833 & -2.0833 & -2.0833 & 0 & 0 & 2.0833 \end{bmatrix}$$

Since all elements are geometrically similar, therefore same stiffness and strain displacement matrix are obtained for each element. When local stiffness matrices are combined a global matrix stiffness is obtained. The nature of this matrix is sparse, therefore here only non-zeros entries will be displayed in table 1 in the appendix.

4.4-Calculating global force matrix

In the given case study, only body forces are acting on the structure, Hence the following local force matrix is obtained for each element by using equation [27].

$$f_b^{(1)} = f_b^{(2)} = f_b^{(3)} = f_b^{(4)} = \begin{bmatrix} 0 & -375 & 0 & -375 & 0 & -375 \end{bmatrix}^T$$

Global force vector obtained by assembling local vector of each element is shown in Appendix 1. Where R_1 to R_6 are the reaction forces that act on the structure as a result of displacement boundary conditions.

4.5-Imposing boundary conditions

After imposing the displacement boundary conditions, global stiffness and force tensors are modified and presented in Appendix 1.

4.6-Solving system of linear equation

Following nodal displacements are obtained when global system of linear equation is solved.

$$\underline{U} = \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \\ u_5 \\ v_5 \\ u_6 \\ v_6 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ -0.000128205128205 \\ -0.001132586632479 \\ 0 \\ -0.003867629367521 \\ 0 \\ -0.010000000000000 \end{bmatrix}$$

4.7-Solving traction forces

When the estimated nodal displacement, traction force vector is calculated using the system of equation we had before imposing the boundary conditions. Using this procedure the reaction traction force vector is calculated by plugging in nodal displacement in equation 16 and following values are obtained,

$$R = \begin{bmatrix} R_{1x} \\ R_{1y} \\ R_{2x} \\ R_{2y} \\ R_{3x} \\ R_{3y} \\ R_{4x} \\ R_{4y} \\ R_{5x} \\ R_{5y} \\ R_{6x} \\ R_{6y} \end{bmatrix} = 1E7 \times \begin{bmatrix} 0.117977774216524 \\ 0.02674901709402 \\ 0.908119658119658 \\ 1.139826139601140 \\ -0.402878059116809 \\ 2.014427795584046 \\ 0 \\ 0 \\ -0.053418803418803 \\ 0 \\ -0.569800569800570 \\ -3.180550836894588 \end{bmatrix}$$

Verification

The given nodal displacement results are verified against results obtained from commercial finite element software ABAQUS. Table 1 shows comparison of results obtained from ABAQUS and the presented work. Relative error obtained for the nodal displacements upon comparison shows are a good agreement of results. Horizontal displacements yielded in ABAQUS for the given case study are displayed in figure 1.

Table 1: Comparison of results from ABAQUS and the presented work

Nodal displacements	Results(m)	Results from ABAQUS(m)	Percentage Relative Error
u_1	0	0	0
v_1	0	0	0
u_2	0	0	0
v_2	0	0	0
u_3	0	0	0
v_3	0	0	0
u_4	-0.000128205128205	-0.00012523	2.32%
v_4	-0.001132586632479	$-\mathbf{0.0011124}$	1.78%
u_5	0	0	0
v_5	-0.003867629367521	-0.0036712	4.851 %
u_6	0	0	0
v_6	-0.010000000000000	-0.01	0

Algorithm

A MATLAB code is generated that is able to given the values of nodal displacements and reaction traction forces for arbitrary number of elements. The algorithm is described in detail in Appendix 2. To check verify the algorithm, the given problem is solved using 9 elements as shown in figure 9 and results of nodal displacement are compared with ABAQUS as given in table 2.

Table 2: Results for 10 node (9 element) domain

Nodal displacements	Results (m)	Results from ABAQUS (m)
u_1	0	0
v_1	0	0
u_2	0	0
v_2	0	0
u_3	0	0
v_3	0	0
u_4	0	0
v_4	0	0
u_5	0	0
v_5	-0.002044929844968	-0.002046
u_6	0	0
v_6	-0.005015455096453	-0.005016

u_7	0	0
v_7	-0.010000000000000	-0.01
u_8	-0.000017646145818	-0.0000180
v_8	-0.002215611824367	-0.00221601
u_9	-0.000252570951408	-0.00025260
v_9	-0.000310339106653	-0.000310341
u_{10}	-0.000393911088497	-0.0003941
v_{10}	-0.001183248903195	-0.001183250

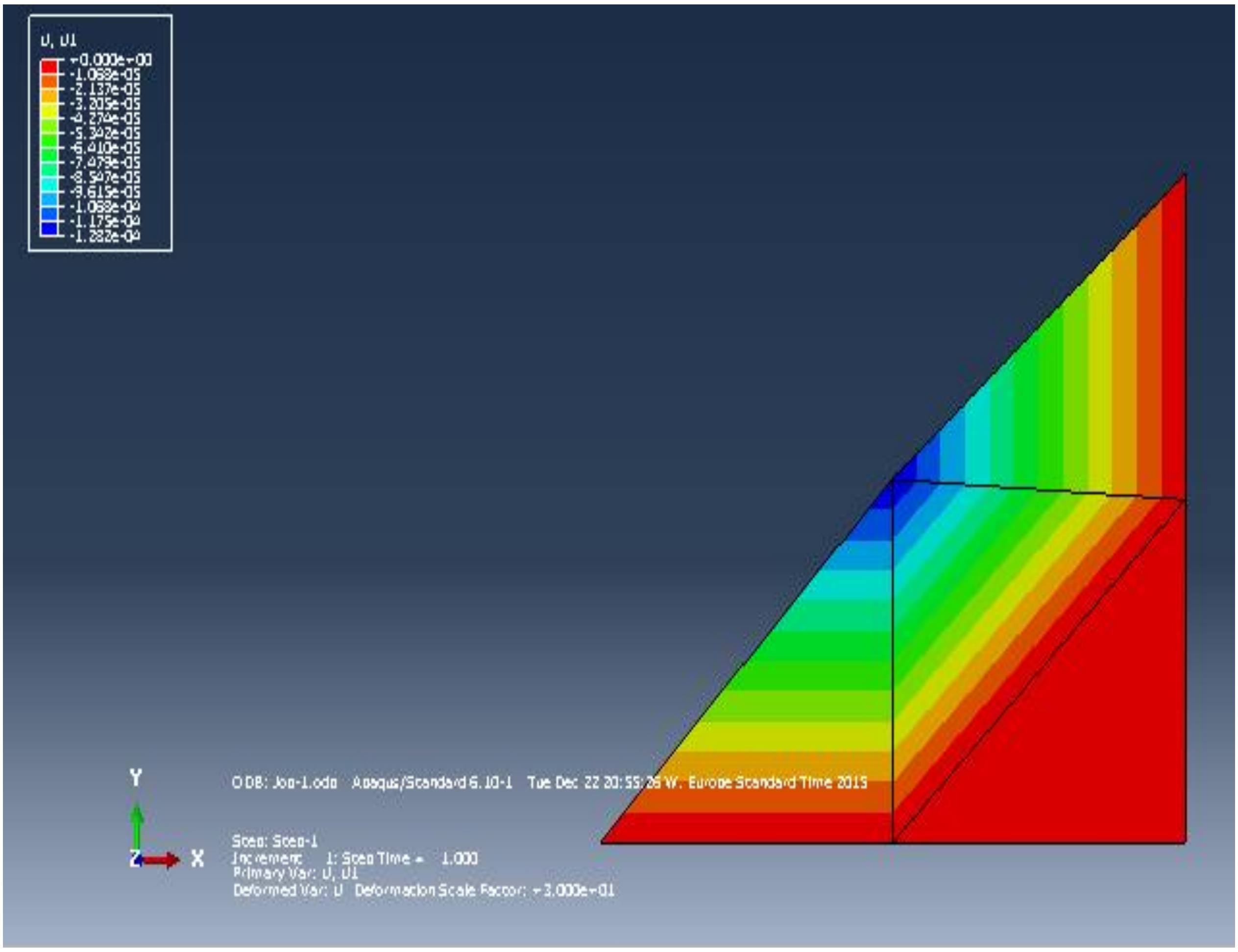

Figure 3: Horizontal displacements obtained from ABAQU

References

- Accessed from [http://www.colorado.edu/engineering/CAS/courses.d/IFEM.d/IFEM.Ch14.d/IFEM.Ch14.pdf], Accessed at [19, 12, 2015]

- Eugenio Onate, Pedro Diez, Francisco Zarate & Astonia Larese, (2008), ‘Introduction to finite element method’ Online access at [19,12,2015]
- Carlos Saracibar, (10, 2015), ‘Lecture 12: Variational Principles’, Polytechnic University of Catalunya UPC.

Appendix No.1

Table 3: Global stiffness matrix before imposing boundary conditions

Entries of global stiffness matrix (Row, Column)	Value *$10E^{10}$ (N/m)

1,1	0.520833333333333
3,1	-0.520833333333333
4,1	0.104166666666667
8,1	-0.104166666666667
2,2	0.208333333333333
3,2	0.208333333333333
4,2	-0.208333333333333
7,2	-0.208333333333333
1,3	-0.520833333333333
2,3	0.208333333333333
3,3	1.458333333333334
4,3	-0.312500000000000
5,3	-0.520833333333333
6,3	0.104166666666667
7,3	-0.416666666666667
8,3	0.312500000000000
10,3	-0.312500000000000
1,4	0.104166666666667
2,4	-0.208333333333333
3,4	-0.312500000000000
4,4	1.458333333333334
5,4	0.208333333333333
6,4	-0.208333333333333
7,4	0.312500000000000
8,4	-1.041666666666667
9,4	-0.312500000000000
3,5	-0.520833333333333
4,5	0.208333333333333
5,5	0.729166666666667
6,5	-0.312500000000000
9,5	-0.208333333333333
10,5	0.104166666666667
3,6	0.104166666666667
4,6	-0.208333333333333
5,6	-0.312500000000000
6,6	0.729166666666667
9,6	0.208333333333333
10,6	-0.520833333333333
2,7	-0.208333333333333
3,7	-0.416666666666667
4,7	0.312500000000000
7,7	1.458333333333334
8,7	-0.312500000000000
9,7	-1.041666666666667
10,7	0.312500000000000
12,7	-0.104166666666667
1,8	-0.104166666666667
3,8	0.312500000000000

4,8	-1.041666666666667
7,8	-0.312500000000000
8,8	1.458333333333334
9,8	0.312500000000000
10,8	-0.416666666666667
11,8	-0.208333333333333
4,9	-0.312500000000000
5,9	-0.208333333333333
6,9	0.208333333333333
7,9	-1.041666666666667
8,9	0.312500000000000
9,9	1.458333333333334
10,9	-0.312500000000000
11,9	-0.208333333333333
12,9	0.104166666666667
3,1	-0.312500000000000
5,1	0.104166666666667
6,1	-0.520833333333333
7,1	0.312500000000000
8,1	-0.416666666666667
9,1	-0.312500000000000
10,1	1.458333333333334
11,1	0.208333333333333
12,1	-0.520833333333333
8,11	-0.208333333333333
9,11	-0.208333333333333
10,11	0.208333333333333
11,11	0.208333333333333
7,12	-0.104166666666667
9,12	0.104166666666667
10,12	-0.520833333333333
12,12	0.520833333333333

Table 4: Global force vector before imposing boundary conditions

Entries of global force vector (Row, Column)	Value (N)
1,1	R_{1x}
2,1	-375 + R_{1y}
3,1	R_{2x}
4,1	-1125 + R_{2y}
5,1	R_{3x}
6,1	-375 + R_{3y}
8,1	-1125
9,1	R_{5x}
10,1	-1125
11,1	R_{6x}
12,1	-375 + R_{6y}

Table 5: Global stiffness matrix after imposing boundary conditions

Entries of global stiffness matrix (Row, Column)	Value x $1E^{10}$ (N/m)
1,1	0.000000000100000
8,1	-0.104166666666667
2,2	0.000000000100000
7,2	-0.208333333333333
3,3	0.000000000100000
7,3	-0.416666666666667
8,3	0.312500000000000
10,3	-0.312500000000000
4,4	0.000000000100000
7,4	0.312500000000000
8,4	-1.041666666666667
5,5	0.000000000100000
10,5	0.104166666666667
6,6	0.000000000100000
10,6	-0.520833333333333
7,7	1.458333333333334
8,7	-0.312500000000000
10,7	0.312500000000000
7,8	-0.312500000000000
8,8	1.458333333333334
10,8	-0.416666666666667
7,9	-1.041666666666667
8,9	0.312500000000000
9,9	0.000000000100000
10,9	-0.312500000000000

7,1	0.312500000000000
8,1	-0.416666666666667
10,1	1.458333333333334
8,11	-0.208333333333333
10,11	0.208333333333333
11,11	0.000000000100000
7,12	-0.104166666666667
10,12	-0.520833333333333
12,12	0.000000000100000

Table 6: Global force vector after imposing boundary conditions

Entries of global force vector (Row, Column)	Value x10E^3 (N)
8,1	-1.125000000000000
10,1	-1.125000000000000
12,1	-0.000010000000000

Appendix No.2

```
clc
clear all
format long
%preproc
ndof = 2 ;
nelem = 4 ;
nnode = 6;
nelnodes = 3*ones(nelem,1);
mesh = [2,4,1 ;
        4,2,5 ;
        3,5,2 ;
        5,6,4]';
% define mesh and topology
X = [-1.5,-1.5,-3;
     -1.5,-1.5 0;
      0,0,-1.5;
      0,0,-1.5];
Y = [0 1.5 0;
     1.5 0 1.5;
     0 1.5 0;
     1.5 3 1.5] ;
%solver
%local stiffness matrix
for i = 1:nelem
[k1(:,:,i),B,E] = stiffmatrix(X(i,:),Y(i,:));
end
% global stiffness matrix
[Stif ] = stiffness(k1,mesh,ndof,nelem,nnode,nelnodes);
% local force vector
f = force();
% global force vector
[F] = globalforce(f,mesh,ndof,nelem,nnode,nelnodes) ;
Ft = F ;
St = Stif ;
% Set of nodes where boundary condition is assigned
nol1 = [1,2,3];
nol2 = [6] ;
nol3 = [5] ;
% Prescribing boundary ccondition
fixnodes=[nol1 ; ones(1,length(nol1)); 0.0*ones(1,length(nol1))];
fixnodes=[fixnodes [nol1 ; 2*ones(1,length(nol1)); 0.0*ones(1,length(nol1))]];
fixnodes=[fixnodes [nol2 ; ones(1,length(nol2)); 0*ones(1,length(nol2))]];
fixnodes=[fixnodes [nol2 ;2*ones(1,length(nol2)); -0.01*ones(1,length(nol2))]];
fixnodes=[fixnodes [nol3 ;ones(1,length(nol3)); 0.0*ones(1,length(nol3))]];
nfix=length(fixnodes(1,:));
 for n=1:nfix
    rw = ndof*(fixnodes(1,n)-1) + fixnodes(2,n);
     for cl=1:ndof*nnode ;
         Stif(rw,cl) = 0;
     end
         Stif(rw,rw) = 1.;
         F(rw) = fixnodes(3,n);
 end
 %solving system of equation
 w = (Stif\F) ;
 %displaying output
 disp(w)
 % Post processing to estimate global traction vector
 r =  Spp*w-Fpp ;
```