Homework 2 FEM

Martí Burcet Rodríguez

22 December 2015

1 Problem Statement:

The problem to solve is the deformation of a thin plate under self-weight and an imposed external displacement at the top of it. The plate has a thickness of one meter, so a plane stress model is chosen. Since the plate is symmetric only the left part of the domain is studied with a discretization of four linear triangles as seen in Figure 1.

Figure 1: Discretization used to study the thin plate deformation.

2 Strong Form:

The first thing that should be done to model any problem using the Finite Elements Method (FEM) is to write down the equations that rule the system in its Strong Form. In the case of plane stress the relation between strain and stresses in linear elasticity is governed by (1).

$$\sigma = D\varepsilon \tag{1}$$

with

$$D = \begin{pmatrix} \frac{E}{1-\nu^2} & \nu\frac{E}{1-\nu^2} & 0 \\ \nu\frac{E}{1-\nu^2} & \frac{E}{1-\nu^2} & 0 \\ 0 & 0 & \frac{E}{2(1+\nu)} \end{pmatrix}$$

and

$$\varepsilon = \bigtriangledown u^S = \frac{1}{2}(\bigtriangledown u + (\bigtriangledown u)^T) \tag{2}$$

The strain tensor is the symmetrized gradient of deformation (2). If we replace (2) in (1) and we impose equilibrium of linear momentum, the general strong form of the problem is given by (3).

$$\bigtriangledown\cdot(D \bigtriangledown u^S) + \rho b = D \bigtriangledown^2 u + \rho b = 0 \tag{3}$$

To reproduce the full model with the reduced domain, some additional consideration have to be done. The symmetry axis corresponding to the bars $\overline{35}$ and $\overline{56}$ is affected by the right and left sides which have opposed horizontal displacement. Therefore this axis won't deform in the x-direction and we need to impose this in the boundary conditions. In addition the three lower nodes are fixed to the ground so they cannot move neither horizontally nor vertically. These conditions are expressed as:

$$\overline{u}(x,0) = 0 \quad ; \quad \overline{v}(3,y) = 0$$

3 Connectivities and Nodal Coordinates:

In order to compute the FEM approximation of the problem, the geometrical properties of the mesh have to be expressed in two arrays: the Nodal Coordinates and Connectivity ones. The first one just contains by rows in the first columns the x-coordinate of the i-th node and in the second one the y-coordinate. Following the global numbering of Figure 1 the Nodal Coordinates matrix comes to be:

$$X = \begin{pmatrix} 0 & 0 \\ 1.5 & 0 \\ 3 & 0 \\ 1.5 & 1.5 \\ 3 & 1.5 \\ 3 & 3 \end{pmatrix}$$

The connectivity matrix contains by rows the relation between the global and local numbering of the i-th element. Following the recommendation of the problem statement, the local element numbering has been chosen to start from the lower right corner of each element and following anticlockwise. This gives the following connectivity matrix for elements from 1 to 4:

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

4 Linear System of Equations:

From the strong form of the problem it is derived the weak form by introducing virtual displacements and integrating by parts. From [1] we obtain which is the expression of the elemental stiffness matrix of one bar of the linear triangular element.

$$K_{ij}^e = \frac{t}{4A^e} \begin{pmatrix} b_i b_j d_{11} + c_i c_j d_{33} & b_i c_j d_{12} + b_j c_j d_{33} \\ c_i b_j d_{21} + b_i c_j d_{33} & b_i b_j d_{33} + c_i c_j d_{22} \end{pmatrix}$$

where

$$b_i = y_j - y_k \quad ; \quad c_i = x_k - x_j$$

This matrix is computed for all the three bars of the element ($\overline{12}$, $\overline{23}$ and $\overline{31}$ in local numbering). Then this can be either assembled into an element stiffness matrix or directly to the global one. In the first case, each bar contribution is placed into the ij-th place using local numbering. After this the local stiffness matrices have to be assembled again into the global stiffness matrix. This procedure is not efficient because requires to store temporally matrices that will finish assembled all together in the global matrix. Instead of doing this, it is much more efficient to compute each of the contributions of the bars and assemble them directly into the global stiffness matrix. This is what is shown in the global stiffness matrix K:

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

Once the global stiffness matrix is assembled, the same has to be done for the force vector. In this particular case the external forces to consider are the self weight of the plate and the applied initial displacement of the top node. The contribution of body forces ($-\rho g$) is given by the next expression (4).

$$f_{bi}^e = \int_\Omega N_i^T \mathbf{b} t dA = \frac{(At)^e}{3} \begin{pmatrix} b_x \\ b_y \end{pmatrix} = \frac{(At)^e}{3} \begin{pmatrix} 0 \\ -\rho g \end{pmatrix} \tag{4}$$

In (4) the nodal contribution of the self-weight is split in equal parts because it produces the same momentum on the element. The imposed vertical displacement in node 6 is introduced as the equivalent force required to produce the displacement (5). In nodes 1, 2, and 3 there will be the vertical and horizontal reactions from the unions, so they have to be introduced as unknowns that will be determined after solving the system of equations.

$$f_{\varepsilon_i}^e = \int_\Omega B_i^T D \varepsilon^0 t dA = \frac{t^e}{2} \begin{pmatrix} b_i(d_{11}\varepsilon_x^0 + d_{12}\varepsilon_y^0) + c_i d_{33}\gamma_{xy}^0 \\ c_i(d_{21}\varepsilon_x^0 + d_{22}\varepsilon_y^0) + b_i d_{33}\gamma_{xy}^0 \end{pmatrix} \tag{5}$$

where

$$\varepsilon_x^0 = \frac{\Delta x}{l_x^e}, \;\; \varepsilon_y^0 = \frac{\Delta y}{l_y^e}, \;\; \gamma_{xy}^0 = \frac{1}{2}\left(\frac{\Delta x}{l_y^e} + \frac{\Delta y}{l_x^e}\right)$$

We note here that in our case the structure is modelled assuming plain stress. This means that the structure can deform both in the x and y directions.

Therefore there are two degrees of freedom per node, what in our discretization of four elements and six nodes make a total of 12 unknowns. The way to compute this using the FEM is to consider a vector of 12 unknowns (6 horizontal displacements and 6 vertical ones) and solve the problem with the same stiffness matrix for the x and y direction. This can be done either considering a block matrix 12×12 or solving two systems of linear equations with same stiffness matrix but different force and unknowns vector. For a question of space we present the force vectors separately.

$$f_x = \begin{pmatrix} f_3^1 + R_{x1} \\ f_1^1 + f_3^2 + f_2^3 + R_{x2} \\ f_1^3 + R_{x3} \\ f_2^1 + f_2^2 + f_3^4 \\ f_1^2 + f_2^3 + f_1^4 + R_{x5} \\ f_2^4 + f_{\varepsilon^0}^4 + R_{x6} \end{pmatrix} f_y = \begin{pmatrix} f_3^1 + R_{y1} \\ f_1^1 + f_3^2 + f_2^3 + R_{y2} \\ f_1^3 + R_{y3} \\ f_2^1 + f_2^2 + f_3^4 \\ f_1^2 + f_2^3 + f_1^4 \\ f_2^4 + f_{\varepsilon^0}^4 \end{pmatrix}$$

As it is known the stiffness matrix of a FEM approximation is singular if the boundary conditions are not imposed. This is done by taking out the rows and columns of the nodes where the displacements (Dirichlet boundary) are already known. Then to consider the effect of that displacement on the remaining nodes, the correspondent component is pre-multiplied by the known displacement and moved to the right-hand side as a force. However in this case since the prescribed displacements are zero, the contribution of each of theses nodes to the force vector will be zero. Regarding the vertical components, the remaining rows and columns are 4, 5, and 6. On the other hand the only node that can deform freely in the horizontal direction is node 4. As the global system is reduced to a 4×4, we present the global system (6).

$$\begin{pmatrix} (K_{11}^2 + K_{22}^3 + K_{11}^4) & 0 & 0 & 0 \\ 0 & K_{22}^1 + K_{22}^2 + K_{33}^4) & (K_{21}^2 + K_{31}^4) & K_{32}^4 \\ 0 & (K_{12}^2 + K_{13}^4) & (K_{11}^2 + K_{22}^3 + K_{11}^4) & K_{12}^4 \\ 0 & K_{23}^4 & K_{21}^4 & K_{22}^4 \end{pmatrix} \begin{pmatrix} u_{x4}^h \\ u_{y4}^h \\ u_{y5}^h \\ u_{y6}^h \end{pmatrix} = \begin{pmatrix} f_2^1 + f_2^2 + f_3^4 \\ f_2^1 + f_2^2 + f_3^4 \\ f_1^2 + f_2^3 + f_1^4 \\ f_2^4 + f_{\varepsilon^0}^4 \end{pmatrix} \quad (6)$$

5 Numerical approximation:

Once the nodal contributions and the system are well defined the only thing to do is replace each expression with the corresponding value and carry out the numerics. Substituting into the equations the given parameters (E=10 GPa, $\nu = 0.2, \delta = 10^{-2}$ m, t=1 m and $\rho g = 10^3 N/m^2$) this is the system to be solved:

$$\begin{pmatrix} -1.15\ 10^{10} & 0 & 0 & 0 \\ 0 & -1.15\ 10^{10} & 3.13\ 10^{09} & -2.08\ 10^{09} \\ 0 & 3.13\ 10^{09} & -1.77\ 10^{10} & 1.04\ 10^{09} \\ 0 & -2.08\ 10^{09} & 1.04\ 10^{09} & -5.21\ 10^{09} \end{pmatrix} \begin{pmatrix} u_{x4}^h \\ u_{y4}^h \\ u_{y5}^h \\ u_{y6}^h \end{pmatrix} = \begin{pmatrix} -1.1250\ 10^3 \\ -1.1250\ 10^3 \\ -1.1250\ 10^3 \\ -5.2084\ 10^7 \end{pmatrix}$$

Once solved the system, the displacements of all the nodes results in the followings:

$$\begin{pmatrix} u_{x4}^h \\ u_{y4}^h \\ u_{y5}^h \\ u_{y6}^h \end{pmatrix} = \begin{pmatrix} -0.0002 \\ -0.0019 \\ -0.0063 \\ -0.0164 \end{pmatrix}$$

References

[1] Eugenio Oñate, Pedro Díez, Francisco Zarate and Antonia Larese. *Inroduction to the Finite Element Method*. Master of Science in Computational Mechanics, 2008.