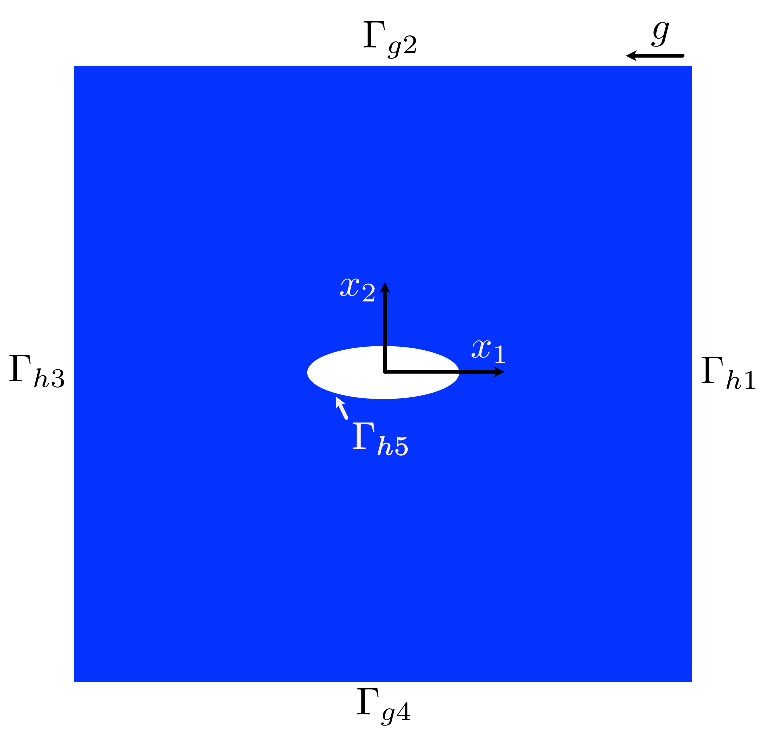
# ME 335A
# Finite Element Analysis
# Instructor: Adrian Lew
# Computing Project #3

May 27, 2023

Due Wednesday, June 7, 2023

In this assignment you will be solving a two dimensional linear elasticity problem on a square domain $(-1, 1) \times (-1, 1)$ with and without a hole. The hole will take the form of an ellipse with a horizontal half-axis of 0.25 and a different smaller vertical half-axis. All lengths are given in $m$. The body is attached to two grips that stretch it from the top and the bottom lines, and the body deforms accordingly. The remaining boundaries are traction free. The labels of the boundaries is indicated in the figure.




The elastic material in this problem is a polymer, with Young's modulus $E$ and Poisson's ratio $\nu$ to be set later, and mass density $\rho = 10\text{kg}/\text{m}^3$. Gravity acts in the horizontal left direction (towards the left of the pdf document), and is given by $g$. The boundary conditions of the problem are then:

$$\begin{aligned} \mathbf{u} &= (0, g_{2y}) && \text{on} \quad \Gamma_{g2} \\ \mathbf{u} &= \mathbf{0} && \text{on} \quad \Gamma_{g4} \\ \sigma \cdot \check{n} &= \mathbf{0} && \text{on} \quad \Gamma_{h1} \cup \Gamma_{h3} \cup \Gamma_{h5} \end{aligned}$$

**Getting Started.** Posted on Canvas you will find two files, `CP3.m` and `CP3Mesh.m`. The first file is the incomplete file that you need to fill in, just as in CP-2. The second file is the custom-made function that creates meshes for the geometry of this problem. It has a single function, `CP3Mesh(HMax, eccentricity)`, where `HMax` is the largest acceptable mesh size, and `eccentricity` is a number between 0 and 1. When `eccentricity`=0, we obtain a square without a hole, and for `eccentricity=1` we get a circular hole. The function then triangulates the domain with $P_2$-triangles, and returns the arrays X, LV, and BE as described in the notes, with some changes described below, and one more array BN. This last array contain the indices of all vertices that lie on the boundary of the domain: BN(1,i) gives a vertex number (so that X(:, BN$(1, i)$) contains its coordinates) while BN(2,i) indicates to what line of the boundary the vertex with such number belongs to according to:

| Value | Line |
|---|---|
| 1 | $\Gamma_{h1}$ |
| 2 | $\Gamma_{g2}$ |
| 3 | $\Gamma_{h3}$ |
| 4 | $\Gamma_{g4}$ |
| 5 | $\Gamma_{h5}$ |

For $P_2$ triangles, LV(:,e) returns the indices of the six nodes of the triangles $e$, with LV(a,e) being the a-th node of element e, with the numbering discussed in class. Similarly, BE(1:3,e) gives the indices of the 3 nodes of an element on the boundary, with BE(3,e) inidicates the index of the middle node. Finally, BE(4,e) indicates the line to which the edge belongs.

1. The first three tasks are:

   (a) (30) Code `[Ke,Fe]=elementKandF(xe,Ee,nue,be)`, which computes the element stiffness matrix and element load vector for a $P_2$ triangular element for elasticity, in which the integration is performed through the 3-point quadrature rule given in class. You also need to code , as well as the section that imposes boundary conditions.

   (b) (5) Code the imposition of the boundary conditions, as prescribed in this problem, after line 47 in `CP3.m`.

   (c) (10) Code `[Strains, Stresses]=computeStrainAndStress(xe,ue,Ee,nue)`, which receives the coordinate of *(only) the three vertices* of a triangle in `xe`, the displacement of the 6 nodes of the triangle in `ue`, and Young's modulus and Poisson's ratio, and returns the stresses and strains tensors (matrices) at the vertices of the triangle. This is for plotting.

   The array `ue(1:6)` returns the $x_1$-components for the six nodes, and `ue(7:12)` the corresponding $x_2$-components. The matrix `Strains(:,:,q)` returns the $2 \times 2$ strain at vertex $q$ of the triangle. Similarly for `Stresses(:,:,q)`.

   This function is used to plot the stressses and strains. You should be able to reuse a lot of code from `[Ke,Fe]=elementKandF(xe,Ee,nue,be)` to do this.

For grading purposes, please submit the code through gradescope.

2. (10) To test your code, we will begin by setting `eccentricity`=0,so that we have a square with no holes. We will also set

$$\begin{aligned} \nu &= 0, \\ E &= 1\text{MPa} \\ g &= 0 \qquad \text{no gravity} \\ g_{2y} &= 0.05. \end{aligned}$$

The exact solution of this problem is that both the strain and the stresses should constant in the domain, and equal to

$$\varepsilon = \begin{bmatrix} 0 & 0 \\ 0 & 0.025 \end{bmatrix}, \qquad \sigma = \begin{bmatrix} 0 & 0 \\ 0 & 0.025\text{MPa} \end{bmatrix}.$$

Please verify that you code can reproduce this deformation. This is similar to what is known as the *patch test* in the finite element literature. Show the plot of $\varepsilon_{22}$ and $\sigma_{22}$ over the mesh (as output from the code, with minor tweaks) for `HMax`=0.2, as well as a plot of the reference and deformed configurations, as output by the code.

3. (10) Next, set

$$\begin{aligned} \nu &= 0.45 \\ E &= 1\text{MPa} \\ g &= 0 \qquad \text{no gravity} \\ g_{2y} &= 1, \end{aligned}$$

and `eccentricity` = 0. This will give you a deformation beyond small strains and small displacements, but it is a interesting to see. Please plot the reference and deformed configurations, and the stress components $\sigma_{22}$, $\sigma_{12}$ and $\sigma_{11}$. Are the traction free boundary conditions on $\Gamma_{h1}$ and $\Gamma_{h3}$ well imposed? Comment on this.

4. (10) Now we will set `eccentricity`=1, and

$$\begin{aligned} \nu &= 0.45 \\ E &= 0.001\text{MPa} \\ g &= 9.81\text{m/s}^2 \\ g_{2y} &= 0, \end{aligned}$$

In this part it is important to pay attention to the units. To get a consistent result, you will need to use all SI units, since otherwise we may be adding Newtons with MegaNewtons. Plot the reference and deformed configurations for `HMax`=0.1. As a reference, the maximum displacement of a point on the left edge should be $\approx -0.15$m.

5. (10) Finally, we will study how the stresses change as we pull from a body with an elliptical hole, as the hole become flatter and flatter approximating a crack (Irwin's solution). To this end, set

$$\begin{aligned} \nu &= 0.45 \\ E &= 1\text{MPa} \\ g &= 0 \\ g_{2y} &= 0.05, \end{aligned}$$

and `eccentricity`$\in \{1, 0.5, 0.1\}$. We will also consider `HMax`$\in \{0.1, 0.05\}$, to visually compare the convergence of solutions. To this end, for each case please plot $\sigma_{22}$ with the $x_2$-axis coming out of the page, so that you can appreciate the height of the stress component (use `view([180, 0])` in MATLAB). Would you trust any of the solutions, or what would you do? Explain.