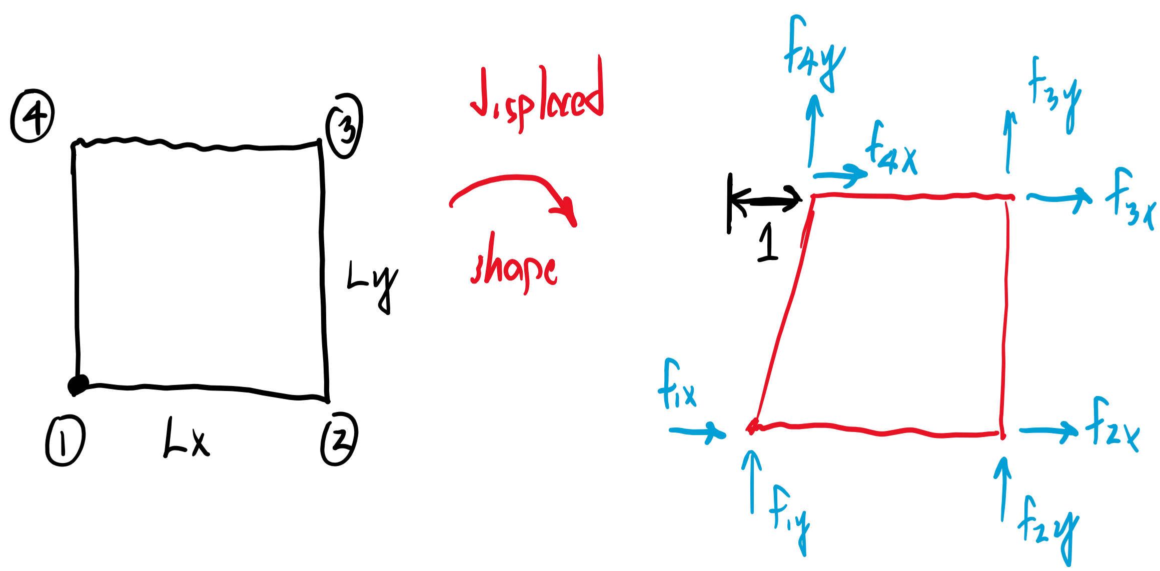
Homework 4

CESG 504 – Finite Element Methods

Due: Wednesday, May 19, 2021

1. ***A Direct-Stiffness Perspective of the 2D Elasticity Stiffness Matrix:*** This problem is intended to be a reminder that the stiffness matrix must satisfy equilibrium. This idea can get lost through all of the steps we took to derive it. Those steps start to feel more mathematical then physical. This question will bring some physical insight back into it.

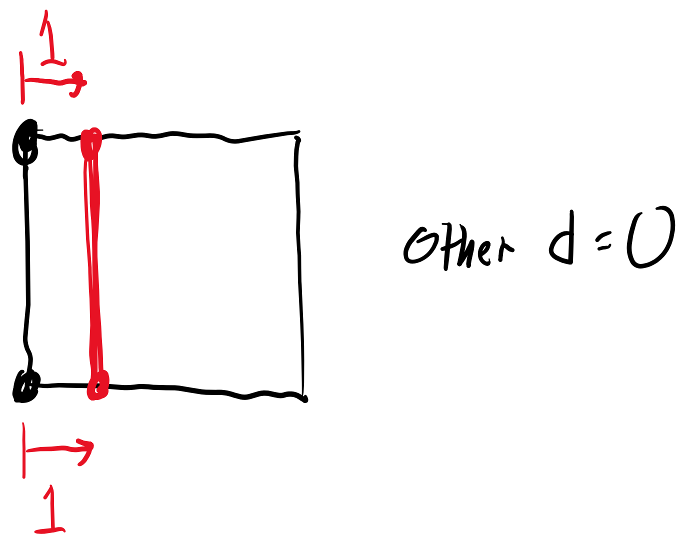
   (a) A 2D rectangular element was subjected to the displacements shown below, i.e., all nodes were held fixed, but node 4 had a unit horizontal displacement applied. Forces are required to sustain this displacement, and those forces can be obtained from the element stiffness matrix. Prove using the 8x8 element stiffness matrix that vertical and horizontal force equilibrium are satisfied by the forces. Note that it should work regardless of what C1 and C2 are, i.e., it will work for plane stress or plane strain. ***Hint:*** You will only need one column of the stiffness matrix.



(b) The moment equilibrium brings up an interesting question. Does the stiffness matrix address the fact that the lever arm of the forces at node 4 depends on the deformation, or does it simply assume the deformations are small enough that no P-δ need be considered? For example, in the deformed configuration f_{4y} has a lever arm about node 1, but it is probably very small. Answer this question by checking equilibrium about node 1. ***Hint 1:*** Check the easy option first, i.e., assume the lever arms are unchanged by the deformation and see if moment equilibrium is achieved in this scenario, this means you can assume that f_{4y} causes no moment about node 1. ***Hint 2:*** You won't have to check the nonlinear option, which makes your work easier, but the downside is the stiffness method we have built is not sufficient for large deformations in its current form (though perhaps you can envision a way to make it work by updating the mesh).

Discussion: You may be wondering how we can say the displacement is small, when we applied a unit displacement. We can do this because the stiffness matrix coefficients all correspond to the force that would be present if the response was linear, and a unit displacement was applied. In practice, we would not apply a unit displacement so we would need to also scale all of the forces by the actual displacement, but since they would all be scaled the same amount there is no change in the outcome. Given that many materials yield at ~2% strain, a more realistic difference in displacement between the left and right nodes would be 2% of Lx or Ly, which is very small.

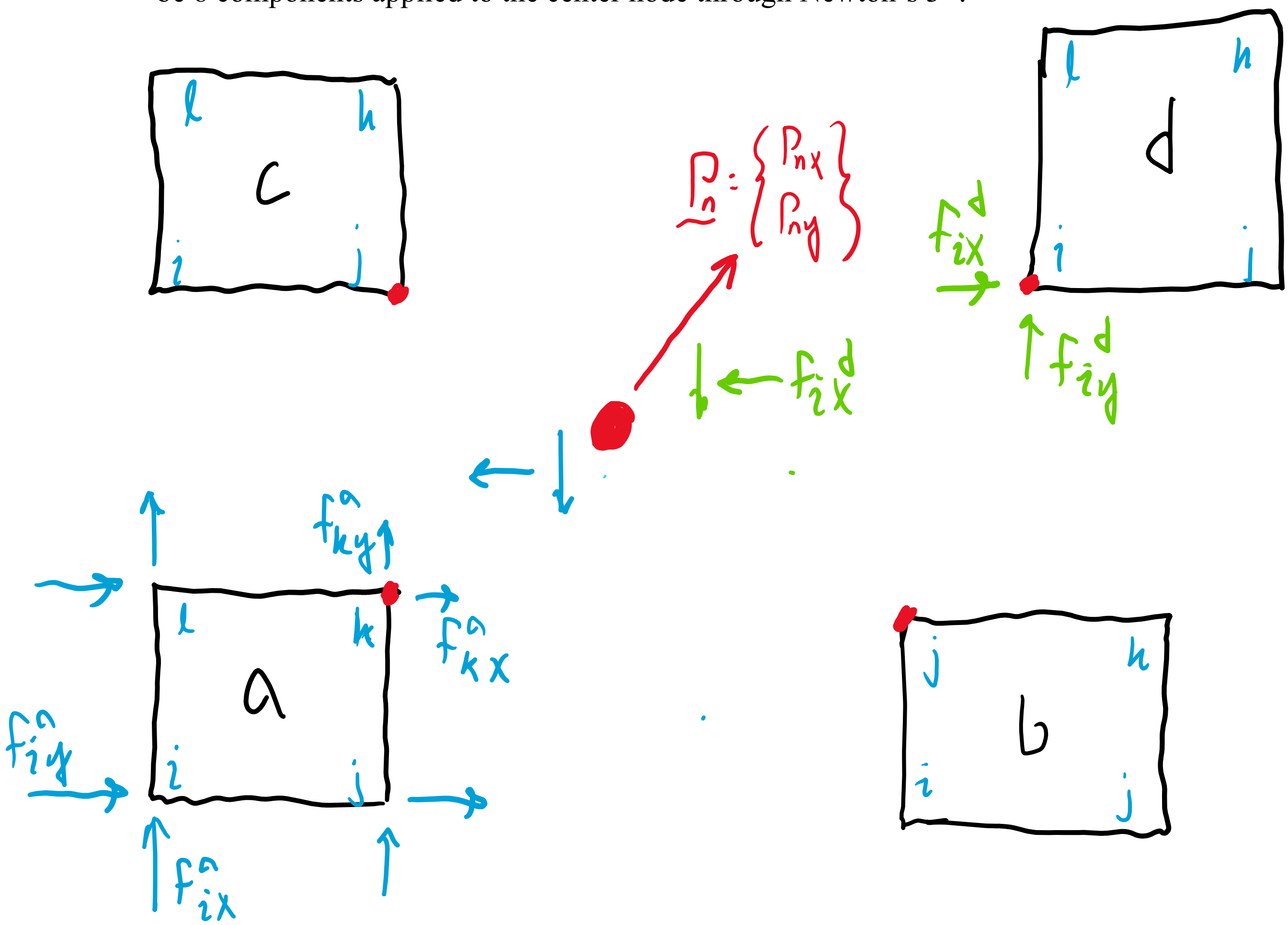
(c) Consider instead the following scenario, where nodes 1 and 4 are both subjected to a unit displacement. Find out what vertical forces are required to sustain this. Once again, the stiffness matrix will give you the result. One might assume that since there is no vertical displacement the vertical reactions would be zero, why is that not the case? ***Hint:*** The answer is in the equation you will find for the vertical force.



(d) Re-derive the 8x8 stiffness matrix for 2D elasticity, but this time, obtain it from the strong form. This means solving $u_x(x,y), u_y(x,y)$ using the strong form of 2D elasticity with boundary conditions given by $u_x(0,0) = d_{1x}$, $u_y(0,0) = d_{1y}, u_x(L_x, 0) = d_{2x}$, etc. for all of the corners, and traction free boundary conditions along the four edges between the corners. Then, after you solve the displacement fields, you can post-process the point loads needed at the corners, which will be f_{ix}, f_{iy} for the four nodes. Even $\{P_{fef}\}$ will appear for free.

Just Kidding, ***do not do this question.*** It would be a very unwieldy infinite series solution. If you are interested in this, definitely take a PDEs class, and a 3D elasticity or continuum mechanics class. The point of this question is that we are no longer able to compare our weak form solution, which we get via FEM, with an exact strong form solution. The approach I describe here is exactly how we derived the stiffness matrix for truss and beam elements (if you took 501, or many other classic stiffness method classes), but the strong forms get too complex from here on forward.

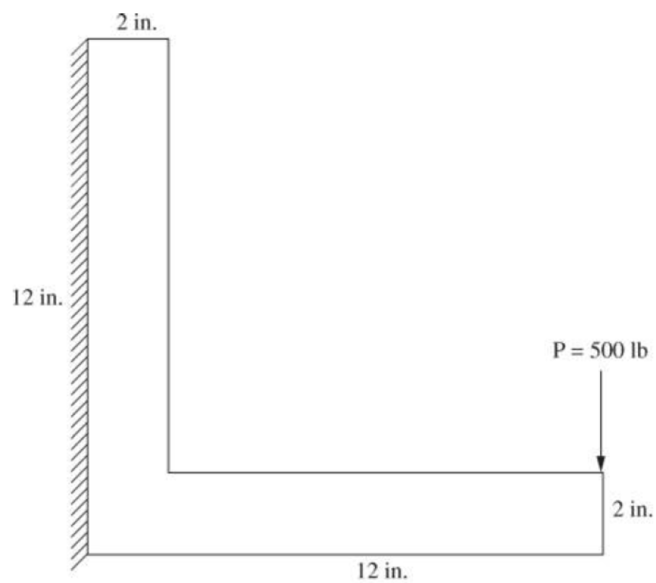
(e) This part looks at the direct stiffness perspective of assembly for 2D elasticity. A node at a junction of 4 rectangular elements is shown below. Fill in the labels on 7 unlabeled arrows below. A few of them are already filled out to give you a sense of what is being asked for. Not nearly all the forces are shown as it gets too cluttered, in total there would be 8 components applied to the center node through Newton's 3rd.



Discussion 1: This question highlights the difference between the global $\{P\}$ vector and its local/element version, which is called either $\{f^e\}$ or $\{P^e\}$. This is the reason for the phrase "assembly is nodal equilibrium" used in direct stiffness, and it applies just as well to 2D elasticity as it does for 1D or beam theory (though we had to cheat by getting the element stiffness matrices via the weak form). Also note that the elements themselves are also in equilibrium, since the $\{f^e\}$ balances as we saw in the previous question. This must be true as the FBD of any cutout whatsoever must be in equilibrium, and that includes both nodes and elements.

Discussion 2: In diffusion problem, the $\{f^e\}$ vectors are instead the element fluxes (but approximated as being "channeled" through the nodes instead of edges), and the $\{P\}$ vector is the net flux into or out of the domain, which balances to be zero unless there is a point source or sink at the node. For torsion problems, 'flux' is always zero on interior.

2. This problem focuses on the ¼” thick bracket shown below. Use Abaqus CAE and 2D modeling to solve this problem. Assume $E = 30{,}000ksi, \nu = 0.3$.



(a) Gradually increase the number of elements, and for each case, plot the maximum principal stress at the re-entrant corner. You will find it does not converge, why?
(b) Add a ½” radius fillet to the corner, and repeat the analysis. Now it should converge.

Discussion: The divergent stress is a real phenomenon for re-entrant corners, not just a numerical issue. However, while it is not ideal (and fillets are used to avoid it), local stress singularities will be handled by yielding (i.e., getting out of the way) so it does not necessarily lead to structural failure. Essentially, the material will yield to achieve an effective fillet. This assumes a ductile material. Note that this kind of stress singularity is the core of the study of fracture mechanics.

3. Using the 2D solver EG3_Cantilever.m as a starting point, explore the validity of "Plane Sections Remain Plane". This code already does a lot of things for you. When you run the code, it shows you a few different outputs (separated by a button press pauses). The last plot shows the deformed shape of what is initially a straight vertical line at L/2. If that line deviates from straight then PSRP is invalid. This is the classic experiment of painting a line on the beam before loading it, and observing what happens.

   (a) Find out at what depth-to-span ratios PSRP loses validity. There is not much to show here, but explore a few different scenarios and describe what you observe. If you can quantify the nonlinearity of the initially straight line in some way, that would be even better. A lot of FEM analysis is about finding ways to characterize the outputs meaningfully.
   (b) Even when PSRP is valid, it is still possible that the Euler Bernoulli assumption that $v' = \theta$ is invalid. This would mean that the rotation of vertical lines is not necessarily the same as the rotation of the neutral axis. This is the +90 part of the full PSRP+90 assumption. The code also calculates the percent error/deviation from the +90 rule, it is output on the screen as the very last output called pE. Create a plot of pE vs the depth to span ratio.
   (c) Summarize the three regimes in terms of depth/L ratios (note it is not really that simple, but will be a good way to characterize the regimes here). The regimes should be (i) thin beams – PSRP+90 valid, this is called Euler Bernoulli beam theory, (ii) intermediate beams – PSRP valid but +90 is not, this is Timoshenko beam theory, (iii) very deep beams, neither approximation is valid, and often "shear struts" are a dominant load transfer mechanism leading to more complex deformation fields.