AME40541/60541: Finite Element Methods Homework 3: Due Friday, March 20, 2020

Problem 1: (25 points) Consider a single one-dimensional, (p + 1)-node Lagrangian element with nodes located as x_1^e, \ldots, x_{p+1}^e .

- (a) What is the polynomial space associated with the element?
- (b) Write the expressions for the element basis functions $\{\phi_1^e, \dots, \phi_{p+1}^e\}$ and their derivatives in terms of the nodal positions.
- (c) Implement a function that evaluates all one-dimensional Lagrange polynomials and their derivatives associated with nodes x_1^e, \ldots, x_{p+1}^e . Starter code provided in Homework 3 code distribution. Check you code: all basis functions should possess the nodal/Lagrangian property; in addition, the basis functions should satisfy

$$\sum_{i=1}^{p+1} \phi_i^e(x) = 1, \qquad \sum_{i=1}^{p+1} \frac{d\phi_i^e}{dx}(x) = 0$$

for any $x \in [x_1^e, x_{p+1}^e]$.

- (d) Plot the basis functions and their derivatives for k = 2, ..., 5 nodes equally spaced in the domain [-1, 1].
- (e) Consider a one-dimensional domain, discretized by 3 of 5-node Lagrangian elements. Create a global numbering for the mesh and write the e2vcg matrix.

Problem 2: (15 points) (AME60541) Construct the weak formulation of Poisson's equation with only natural boundary conditions. Be sure to explicitly state the trial and test space. Show the bilinear form is not coercive over the trial space.

Problem 3: (15 points) (AME60541) Prove that a symmetric, coercive bilinear form leads to a finite element stiffness matrix that is symmetric positive semi-definite.

Problem 4: (30 points) From S. Govindjee, UC Berkeley. In this problem you will examine the behavior of linear vs quadratic elements in trying to estimate the stress around a hole in a plate.

- 1) Start COMSOL: 2D analysis, structural mechanics, stationary analysis.
- 2) Under Global Definition and Parameters
 - Right-click *Parameters* and define a new parameter named *mesh_size* (under *Name*). We will be varying the *Expression* value; set it to 0.05 for now.
- 3) Create the geometry as follows:
 - Rectangle: $0 \le x \le 10$ m and $0 \le y \le 2$ m
 - Circle: center (5, 1)m and r = 0.5m
 - Take the difference of the rectangle and the circle (using *Booleans and Partitions, Difference* under the *Geometry* tab) to create the domain shown in the figure. If you have trouble selecting the appropriate object when creating the difference, use the middle button to cycle through objects while hovering the mouse over them.



• Add a point at (x, y) = (0, 1) for later use

Our objective is to compute the maximum von Mises stress $\max \sigma_{mises}$ throughout the domain using different mesh densities and element types.

4) Within Solid Mechanics set the 2D Approximation to Plane Stress and the Thickness to 1 mm. Also set the Discretization to Linear. If you do not see Discretization in the Solid Mechanics settings, you can add it through the Model Builder window.



- 5) Set the elastic properties as: Young's modulus = 70 GPa and Poisson's ratio = 0.33.
- 6) Add a *Prescribed Displacement* and a *Prescribed Load*. On the left edge, fix the displacement in the x-direction to be zero. At the point (0, 1), fix the displacement in both directions. On the right edge, set the load type to *Total Force* with a value of 1 kN in the x-direction.
- 7) Define a User-controlled mesh with Maximum element size set to be mesh_size (your parameter). This forces the mesh to have elements roughly the size of the parameter value.
- 8) Right-click Study and add a Parametric Sweep. Left-click Parametric Sweep and add mesh_size as a parameter name using the plus sign and in the Parameter values field enter: 1 0.8 0.6 0.4 0.2 0.1 0.09 0.08 0.07 0.06 0.05.
- 9) If you now solve (*Compute* under *Study*), COMSOL will solve the problem using the sequence of meshes you defined.
- 10) The maximum von Mises stress happens at the top and bottom of the hole. To extract the needed information, use the *Derived Values* option with a *Point Evaluation*. To get the correct variable in *Point Evaluation* change *Expression* to *solid.mises* by typing it in or using the menus. Make sure the *Data set* is *Study1/Parametric Solutions 1*. If you now *Evaluate All (Results tab)*, it will create a table of values for your for each mesh density.
- 11) Repeat this study with quadratic elements. Go back to *Solid Mechanics* and change the *Discretization* to *Quadratic*. Also go to the *Mesh* tab and change the size to 2**mesh_size*. This will keep the number of nodes close to that of the linear case (more fair comparison). Go to *Study* and *Compute* a new solution. Re-evaluate your *Point Evaluation*. This will add a new column to your results table.
- 12) Export the table with both columns (linear and quadratic elements) and create a plot of mesh size vs. maximum von Mises stress (use a semi-logx plot). Don't forget that your quadratic elements have elements roughly twice the size of your linear elements (even though it won't be reflected in your table). Comment on the meaning of what you observe. Also export a plot of the contours of the von Mises stress over the plate.

Problem 5: (30 points) From S. Govindjee, UC Berkeley. Consider the two-dimensional beam subject to a transverse sinusoidal load below.



Let u(x, y) denote the x-displacement and v(x, y) the y-displacement. The exact solution of the y-displacement along the centerline of the beam with the boundary conditions

$$u(0,0) = v(0,0) = v(L,0) = 0, \qquad q(x,c) = q_0 \sin\left(\frac{\pi x}{L}\right),$$

and the geometric condition $L \gg c$, is

$$v(x,0) = \frac{3q_0L^4}{2c^3\pi^4E}\sin\left(\frac{\pi x}{L}\right)\left[1 + \frac{1+\nu}{2}\frac{\pi c}{L}\tanh\left(\frac{\pi c}{L}\right)\right].$$

This analytical solution was derived under the plane stress assumptions. Select reasonable values for the geometry (the L/c ratio should be at least 10) and material properties (stiffness E and Poissson ratio ν).

- (a) What is the analytical solution at (x, y) = (L/2, 0) for the parameters you chose?
- (b) Model this system in COMSOL. Be sure to use the plane stress assumption with a thickness of t = 0.001Land apply boundary conditions exactly as specified above i.e., do not fix the displacements along an entire edge. For the discretization, consider both linear triangles and quadrilateral elements on a sequence of at least four meshes each of increasing refinement. For each mesh, compute the solution and output v(L/2, 0).
 - Make sure to output the vertical displacement rather than the total displacement, which is the default.
 - Model the domain as two rectangles, one over $[0, L] \times [-c, 0]$ and one over $[0, L] \times [0, c]$. Some versions of COMSOL with have trouble generating a quadrilateral mesh if an isolated point, e.g., at (L/2, 0), exists.
 - COMSOL does not allow you to take the sine of a number with *units*. Therefore, to prescribe the distributed load $\sin(\pi x/L)$, this must be entered as $\sin(pi*(x/L[m]))$ if you are working in units of meters. Also, be sure to apply the load per unit length to be consistent with the assumptions under which the analytical solution was derived.
- (c) On a single figure, plot v(L/2, 0) versus the number of elements for both mesh sequences. Also include the exact solution as a horizontal line (since it does not depend on the number of elements). What do you observe about the accuracy of triangular vs. quadrilateral elements for bending problems?