MAE671 ASSIGNMENT SET
Spring 2008

FEA #1: Elementary one-element problems

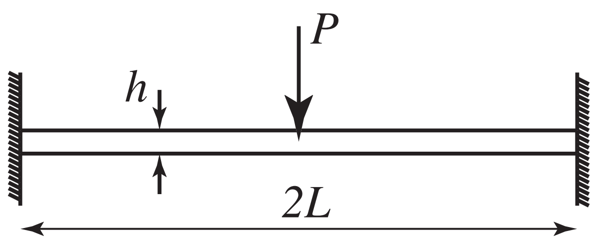
1.1 Create a finite element model of thin plate loaded in tension, consisting of one four-noded quadrilateral element: verify the solution using elementary elasticity theory (i.e. show that the displacements, stresses and strains from the FEA agree with closed-form solutions).

1.2 Create a finite element model of a long, rectangular prism that is compressed in one direction, consisting of one four-noded quadrilateral element: verify the solution using elementary elasticity theory (i.e. show that the displacements, stresses and strains from the FEA agree with closed-form solutions).

1.3 Create a finite element model of a long-rectangular prism that is compressed in two directions (use one four-noded quadrilateral element): verify the solution using elemenatry elasticity theory (i.e. show that the displacements, stresses and strains from the FEA agree with closed-form solutions).

FEA #2: Quasi-2D Problems: beams

Analyze a clamped-clamped beam with a point-load in the center. Using the following properties: Young's



modulus = 100 GPa, Poisson's ration = 0.3, height = 0.05 m, and width = 0.025 m.

2.1 Compare the effect of different element types (i.e. interpolation functions) by determining the minimum number of elements required to obtain results that are within 5% of the analytical solutions (for small deflections) for (a) two-node elements with linear interpolation, and (b) two-node elements with cubic interpolation. Repeat the analysis for three different beam lengths: L/h = 5, L/h = 25 and L/h = 125.

2.2 Illustrate the effect of *large deflections* by determining the load-point displacement at which the corresponding load (to achieve that displacement) is 25% from the analytical solution for small deflections. Demonstrate the effects of mesh density by plotting the critical load-point displacement (at which the linear solution is no longer accurate) versus element size. Repeat the analysis for two different beam lengths: L/h = 25 and L/h = 125.

2.3 Illustrate the effect of *pre-tension* in the beam by prescribing a temperature change to the beam: show that the effect of pre-tension is consistent with the analytical solution for a pre-stressed beam detailed in: Seker, E., Gaskins, J.T., Zhong, J., Bart-Smith, H., Reed, M.L., Kelly, R., Zangari, G. and Begley, M.R. (2007) The effects of post-fabrication annealing on the mechanical properties of free-standing nanoporous gold structures, *Acta Materialia,* **55**, 4593-4602. (The solution in this paper is an asymptotic expansion for small displacements derived from full solution for a point-loaded beam, given in the following: Begley, M.R. and Barker, N.S. (2007) Analysis and design of kinked (bent) beam sensors, *Journal of Micromechanics and Microengineering*, **17**, 350-357.)

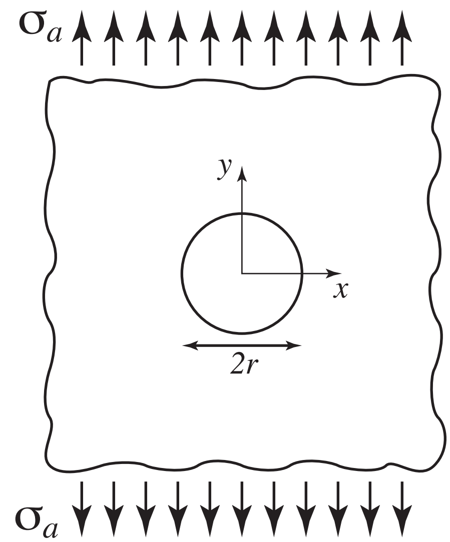
FEA #3: 2-D Problems: plate with hole

Analyze the problem of a circular hole in a thin plate (e.g. plane stress) that is loaded in tension at the top. For an infinite plate (i.e. width and height that are much larger than the hole) the analytical solution is given below.

3.1 Determine the mesh that is needed to accurately capture the stress in the plate; your goal is to obtain a solution that is within 5% of the infinite plate solution, for distances that are less than five times the hole radius. (This should require far less than 500 elements.) Prove convergence by plotting the stress distributions along the mid-plane of the plate for both the FEA and analytical solutions.

3.2 Compare the performance of four-noded and eight-noded quadrilaterals, and identify the coarsest mesh that produces the correct value (i.e.. within 5%) of the maximum tensile stress near the edge of the hole. illustrate the differences between different stress quantities (i.e. elemental averages, stresses at the nodes, etc.) by plotting several curves with each type of calculated stress: superimpose the analytical solution over the numerical results.

3.3 Use FEA to determine the stress concentration factor for the problem of a hole in a plate of finite width, where the hole diameter is 80% of the width of the plate. Illustrate that your mesh is convergent (i.e. that you have the right answer.



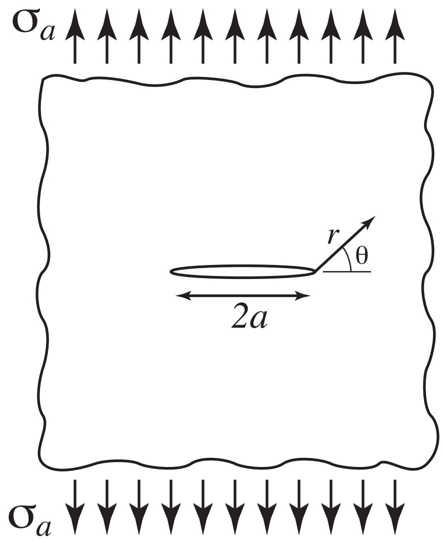
$$\sigma_{yy}(x,0) = \frac{\sigma_a}{2}\left(2 + \left(\frac{a}{x}\right)^2 + 3\left(\frac{a}{x}\right)^3\right)$$

$$\sigma_{xx}(0,y) = \frac{\sigma_a}{2}\left(\left(\frac{a}{y}\right)^2 - 3\left(\frac{a}{y}\right)^3\right)$$

FEA #4: Singularity problems: point-load half-space & crack problems

Analyze the two problems below, and illustrate that the FE analysis yields the given closed-form solutions: to do this, you must (1) determine the size of the mesh that approximates an elastic-half space (i.e.. something with semi-infinite dimensions), and (2) demonstrate that the FEA stress distributions agree with the analytical solutions for locations outside an arbitrary radial distance that is dictated by the mesh distribution. Comment on the difference in convergence with respect to stresses and displacements.

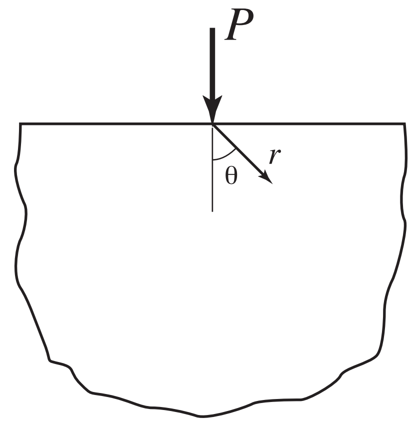
Hints: use log-log plots to illustrate agreement in terms of the power-law dependence of the spatial distribution. Section 3 in the following reference discuss meshing considerations for crack problems (which are also applicable to point-load singularities, I think): Begley, M.R. and Begley, C.J. (2003) Interfacial debonding around rectangular features in multi-layer structures, Special Issue on the Mechanics of Interfaces, *Interface Science* , **11**, 319-327.



$$\sigma_{ij}(r,\theta) = \frac{K_I}{\sqrt{2\pi r}} f_{ij}(\theta)$$

$$f_{11}(\theta) = \cos(\theta/2)\{1 - \sin(\theta/2)\sin(3\theta/2)\}$$
$$f_{12}(\theta) = \cos(\theta/2)\{\sin(\theta/2)\sin(3\theta/2)\}$$
$$f_{22}(\theta) = \cos(\theta/2)\{1 + \sin(\theta/2)\sin(3\theta/2)\}$$

$$u_2(\theta = \pi) = \frac{(\kappa+1)K_I}{2\mu}\sqrt{\frac{r}{2\pi}}$$



$$\sigma_{rr} = -\frac{2P}{\pi r}\cos\theta$$

$$\sigma_{\theta\theta}, \sigma_{r\theta}, \sigma_{zz} = ?$$

$$u_r(\theta = 0) = -\frac{2(1-\nu^2)}{\pi E} P \log r + C\cos\theta$$

FEA #5: Axi-symmetry, material and geometry non-linearity: elastic-plastic cavitation

Analyze the problem of a spherical hole in a block of an elastic-plastic material that is loaded in uniform remote tension in all directions. If the block is large enough (such that the outer boundaries are not sufficient “close” to affect the solution near the hole), the problem is spherically symmetric, implying that the only displacements are radial displacements: i.e. displacements occur along purely radial trajectories. The problem can be addressed by changing the “hole-in-plate” models to axisymmetry, i.e. by modeling a spherical hole in a cylinder.

Background: Since the (deviatoric!) stresses are highest at the hole, this region next to the hole undergoes plasticity before the remote regions of the block, which remains elastic.. As the remote tension from the hole increases, the size of the plastic zone surrounding the hole increases, and hole gets bigger. At a critical value of the remote applied stress, the hole grows uncontrollably at constant load, a phenomenon known as cavitation. Physically, this corresponds to the hole “popping” - in a sense it explodes. This is not an academic/theoretical postulation - it has been observed in experiments (notably spherical inclusions ahead of a crack tip and small voids in ductile fibers used to reinforce ceramics). A simple explanation is that the stored elastic energy in the majority of the block can no longer be “held in check” by the plastic dissipation near the hole: what that happens, plasticity is no longer effective in alleviating stress (and lowering stored elastic energy).

To model this phenomenon, we merely have to include non-linear strain-displacement relationships, and non-linear material descriptions: in the analysis, the remote load is increased while the size of the hole (and plastic zone size) are monitored. When the hole starts growing at a rapid rate (e.g. it doubles in size for very small increase in remote load), cavitation has commenced and the limit load (or cavitation stress) has been reached.

Perhaps surprisingly, the symmetry of the problem implies that a quasi-analytical solution can be found quite easily. By this I mean we can derive an expression that relates the current hole radius to the applied stress - this expression involves evaluating an integral that can not be done in closed form. It can, however, be integrated numerically - hence, it's an analytical approach with a quick numerical computation at the end. If time permits, this approach will be outlined in class. Finally, you can read more about this problem and compare your solutions with those presented in: "Cavitation instabilities in elastic-plastic solids," Y. Huang, J. W. Hutchinson & V. Tvergaard, *Journal of the Mechanics and Physics of Solids*, **39**, 223-241 (1991).

Implementation: To analyze the problem with FEA, we can use the mesh from part 2. We simply need to change the material properties to include non-linear material behavior. Since the problem will be non-linear, we'll need to solve the finite element equations incrementally (i.e. we no longer have a linear set of equations). This is accomplished by making the following changes. Replace the elastic material properties with the following lines:

```
*MATERIAL, NAME=ALUM
*DEFORMATION PLASTICITY
70e9,0.3,100e6,5.0,0.42
```

The numbers on the last line are, respectively: Young's modulus, E, Poisson's ratio, ν, yield stress, σY, hardening exponent, n, and the coefficient of the non-linear term, α. These properties correspond to a tensile test whose stress-strain relationship is: $\varepsilon = \sigma/E + \alpha(\sigma/\sigma_Y)^n$.

To solve the problem iteratively, we need to tell ABAQUS to attempt the solution in small increments. The proper lines are:

```
*STEP,INCR=100
*STATIC
0.05,1.0,1.e-06,0.1
```

The maximum allowable number of increments is dictated by the INCR=100 flag. The first number on the last line is the percentage of the total load that is applied in the first increment. The second number is the final percentage of the total load that we want to reach. (If we set this number to 2.0, we would actually end up applying twice the load that we specify on the load card - in the vast majority of cases, we set this number to one.) The third number is the minimum increment size that we want to allow. ABAQUS will adjust the increment size automatically to speed convergence - it will take large steps when the solution is linear (or nearly so - as during the initial deformation stages when behavior is elastic) and small steps when the equations are highly non-linear (such as when plasticity starts to dominate). The reason for setting a minimum is this: if ABAQUS has trouble finding a solution and takes really small increments (say at the limit we've set at 1.e-6), it would take a million steps (or solutions!) to get our result. If this happens, something is wrong and we should take a look at what's going on. The final number is the maximum step size.

If all we cared about was the final solution, that is, the solution that corresponds to the maximum load, we'd set this value to 1.0 - ABAQUS would get there as quickly as possible. Very often, however, we'd like to know the solutions that are found along the way - this solutions correspond to fractions of the total load. Hence, in one iterative procedure, we find solutions not just for the load we prescribed, but for increasing fractions of the load as well. By limiting the step size, we can make sure that ABAQUS outputs results at multiple loads, rather than just racing to the finish.

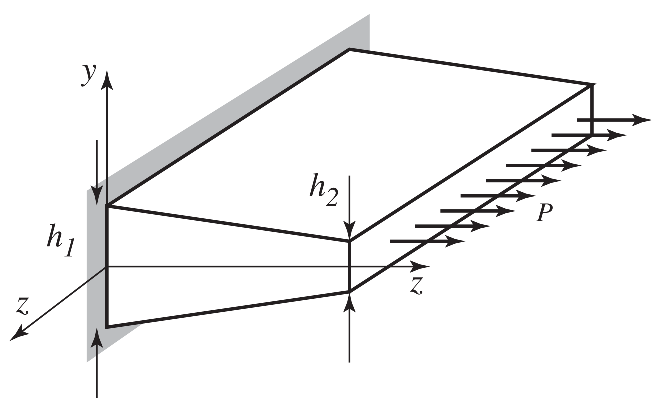
5.1 Solve the elastic-plastic cavitation problem, assuming small geometry changes. (I.e., use the exact *STEP definition outlined above.) Use the mesh you decided was accurate in part 2. The goal is to determine the radius of the hole as a function of applied load - note that you only need to run one analysis! You can monitor the radius of the hole by recording the appropriate displacements at the edge of the hole. Create a plot that plots the hole radius versus applied stress (plot stress on the y-axis) for two different strain hardening exponents. Your final load should be about 10 times the initial yield stress of the material, which is inputted on the material card outlined above. Plot the deformed shape of the mesh at the maximum load. NOTE: this deformed shape may not be realistic!

5.2 Solve the cavitation problem again, this time accounting for large geometry changes. This is accomplished simply by adding the term NLGEOM to the *STEP definition, as in "*STEP,INCR=100,NLGEOM". Again record the hole radius as a function of applied load. As weird as this sounds, you may apply loads that are high enough to crash the analysis! This corresponds to the fact that the radius wants to grow uncontrollably. You'll know the analysis has crashed if it hasn't reach the total solution percentage, i.e. 1.0; this is most easily detected by looking at the *.STA file that ABAQUS creates - this is the status file that lists the increments and increments sizes for which solutions have been found. When you plot the hole radius via applied load, you'll see why the code crashed - the cavitation limit had been reached. Again, run the analysis for two values of strain hardening coefficient, and determine the maximum load that can be applied.

Create one plot comparing the results of your analyses. The plot should have four curves, two curves corresponding to each strain hardening coefficient - one for the small geometry change solution and one for the large geometry change solution. Write a brief paragraph explaining your results and stating the cavitation limit (maximum applied stress) for each case. NOTE: the cavitation limit may be difficult to identify for the small geometry change solution!

THEORY 1: 3 keys to solid mechanics & 1D FEA

T1.1 Combine the three keys of solid mechanics to derive an approximate solution for the *displacement distribution* in the tapered plate illustrated below: the width in the z-direction is much larger than any other dimension. Assume a one-dimensional, one-component stress distribution for the center of the plate, i.e. the only non-zero stress is the direct stress in the x-direction.



T1.2 Illustrate that the one-stress component solution to *T1.1*, while entirely reasonable and fairly accurate, mathematically violates one physical law (equilibrium), one experimentally observed relationship (Hooke's law) and one (at least) boundary condition of the problem (Cauchy's relation).

T1.3 Design your own statically indeterminant 2-D truss problem with at least three bars, three unconstrained degrees of freedom, and a force applied to a single node at an arbitrary angle, and derive the linear equations governing the motion of the unconstrained displacements using the three keys of solid mechanics. Express these equations in the matrix form [K][u]=[F], where [K] is the system's stiffness matrix, [u] is the vector of unknown displacements, and [F] is the vector of applied forces Solve for the displacements for a specific numerical case using Matlab or the like.

THEORY 2: Fundamental Elements of Continuum Mechanics

T2.1 Using force equilibrium on a rectangular differential element, derive the 2-D equilibrium equations in terms of the stress and acceleration components.

T2.2 Using force equilibrium on a “cut block”, derive Cauchy’s relation, which relates stress components at a point to the tractions acting on an arbitrary plane.

T2.3 Using Cauchy’s relation, derive the stress-transformation equations that determine how stresses are converted from one coordinate system to another. Illustrate how Cauchy’s relationship can be used to define an eigenvalue problem that yields the principle stresses and directions, using matrix representations.

T2.4 Using vector algebra, derive the non-linear strain-displacement relationships which account for large displacement gradients.

THEORY 3: Energy principles, Virtual Work, and FEA formulation

T3.1 Using energy principles and variational calculus, derive the governing general equations and boundary conditions for a beam that undergoes axial elongation and Bernoulli-Euler bending: for each boundary condition, illustrate how the variational procedure indicates the choice of boundary conditions. (For example, you can specify the slope at given location, or the moment, but not both.).

T3.2 Solve for the downward deflection of a cantilevered beam with uniform distributed loading using the same functional used for problem T3.1 and the Rayleigh-Ritz method. (This involves assuming a form of the solution and solving for the constants.) Verify the accuracy of your approximate solution using classical textbook solutions for small deflections.

T3.3 Using energy principles and variational calculus, derive the FEA equations for a beam in bending (neglecting axial elongation): clearly identify appropriate nodal variables, the requisite shape functions, the elemental stiffness matrix, nodal force matrix (vector), etc.

T3.4 Using the Principle of Virtual Work (PVW) derive the governing equations and boundary conditions for a 2-D elastic solid. Explain how your derivation indicates you can prescribe either a force or displacement, but not both. Index notation is preferred, but not necessary. To complete this problem, you will (likely) do the following (at some point, the order/appearance of which depends on your algebra): (a) use the conventional small-displacement strain-displacement relationship, (b) note and show that the variation of a derivative is equal to the derivative of a variation, (c) apply the divergence theorem, and (d) integrate by parts.

T3.5 Using the principle of virtual work, derive the FEA equations for a three-noded constant strain triangle. You do not need to explicitly evaluate the entries of all matrices: but you should identify nodal variables and elemental property matrices, including their dimensions, and describe how the individual matrix entries are computed.