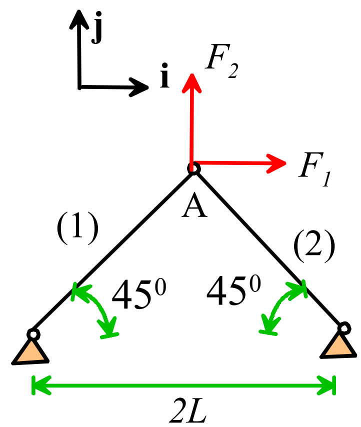
School of Engineering
Brown University

EN234: Computational methods in Structural and Solid Mechanics

Homework 6: FEA for nonlinear materials
Due Fri Oct 27, 2017

1. In this problem you will implement a simple reduced-dimensional model to illustrate how the Newton-Raphson method is used to calculate deformation in a structure with nonlinear force-extension response. The figure shows a simple 2D truss subjected to forces $[F_1, F_2]$ at its apex (A). Suppose that the two members have a nonlinear force-extension relation of the form



$$\frac{T_\alpha}{T_0} = \begin{cases} sign(\varepsilon_\alpha)\left[\sqrt{\dfrac{1+n^2}{(n-1)^2} - \left(\dfrac{n}{n-1} - \dfrac{|\varepsilon_\alpha|}{\varepsilon_0}\right)^2} - \dfrac{1}{n-1}\right] & |\varepsilon_\alpha| \le \varepsilon_0 \\ sign(\varepsilon_\alpha)\left(\dfrac{|\varepsilon_\alpha|}{\varepsilon_0}\right)^{1/n} & |\varepsilon_\alpha| \ge \varepsilon_0 \end{cases}$$

($\alpha = 1,2$), where T_0, ε_0, n are constants and $\varepsilon_\alpha = \Delta l_\alpha / L_\alpha$ is the (infinitesimal) uniaxial strain in each member, and sign(x) = -1 if $x<0$ and sign(x)=+1 if $x>0$

1.1 Suppose that the joint at A experiences an infinitesimal displacement $[u_1, u_2]$. Find the **B** matrix that relates the change in length of the two members to the displacement through

$$\begin{bmatrix} \Delta l_1 \\ \Delta l_2 \end{bmatrix} = \mathbf{B} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix}$$

1.2 Show that the principle of virtual work (for a structure) requires that the tensile internal forces in the two members $[T_1, T_2]$ satisfy

$$\begin{bmatrix} F_1 \\ F_2 \end{bmatrix} - \mathbf{B}^T \begin{bmatrix} T_1(\Delta l_1) \\ T_2(\Delta l_2) \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix}$$

1.3 This equation can be solved for $[u_1, u_2]$ by Newton-Raphson iteration. The procedure (as usual) is to start with an initial guess for the solution $[w_1, w_2]$, and repeatedly correct the guess until the equilibrium solution has been found. The corrections $d\mathbf{w}$ are found by solving a system of linear equations

$$\mathbf{K} d\mathbf{w} = \mathbf{r}$$

Find an expression for the consistent tangent matrix **K** in terms of **B** and the material tangent matrix

$$\mathbf{D} = \begin{bmatrix} dT_1 / d\Delta l_1 & 0 \\ 0 & dT_2 / d\Delta l_2 \end{bmatrix}$$

1.4 Implement this procedure in Matlab. Test your code by plotting the variation of the horizontal and vertical displacement of joint A as functions of F_1 (or both force components, if you are curious).

As a solution to this problem please upload solutions to 1.1-1.3, as well as the graph for 1.4 as a pdf (or equivalent) (the pdf can also include your solutions to 2.1, 2.3 below), and upload your MATLAB code for 1.4 separately. You can choose any sensible set of material parameters to test your code. Note that you can check your Newton-Raphson solver easily by calculating the force-displacement curve by hand (if you prescribe displacements instead of forces, the stretch of each member is known; you can calculate the tensions; and get the forces from equilibrium). You can use any sensible set of material constants for the test – tests with n close to 1 and n of order 10 usually illustrate behavior in the linear and highly nonlinear limits (but be careful not to apply forces that are too big with large n).

2. In practice, any FEA code you might use in the future will have a Newton-Raphson nonlinear solver built-in, so you aren't likely to have to code your own (although you will often need to use Newton-Raphson iterations to solve nonlinear equations inside your element or user material subroutines). To analyze behavior of a nonlinear material, you will only need to calculate the stress and material tangent stiffness matrix. In ABAQUS, this can be accomplished in a UMAT user-subroutine. In this problem you will write an ABAQUS user-material subroutine (UMAT) to implement your version of the simple small-strain 'porous elasticity' material model that is built-in to ABAQUS.

In the simplest version of the model, the stress-strain relation for the material has the form

$$\sigma_{ij} = 2Ge_{ij} + \frac{p_t}{3}\left[1 - \exp\left\{-\frac{1+e_0}{\kappa}(J-1)\right\}\right]\delta_{ij}$$

where G is the shear modulus, κ is the (dimensionless) logarithmic bulk modulus, e_0 is the initial void ratio, p_t is the elastic tensile limit (in the sense that $J \to \infty$ when $\sigma_{kk}/3 \to p_t$), $e_{ij} = \varepsilon_{ij} - \varepsilon_{kk}\delta_{ij}/3$ is the deviatoric strain, and J is the determinant of the deformation gradient. For a strict small strain analysis we can use the approximation $J-1 \approx \varepsilon_{kk}$.

Note that at zero strain, the bulk modulus is related to the Poisson's ratio by

$$\kappa = p_t \frac{1-2\nu}{(1+\nu)G}(1+e_0)$$

2.1 Show that, for small strains, the material tangent matrix can be approximated as

$$\frac{\partial \sigma_{ij}}{\partial \varepsilon_{pq}} = G(\delta_{ip}\delta_{jq} + \delta_{jp}\delta_{iq}) - \frac{2G}{3}\delta_{ij}\delta_{pq} + \frac{p_t}{3}\frac{1+e_0}{\kappa}\left[\exp\left\{-\frac{1+e_0}{\kappa}\varepsilon_{kk}\right\}\right]\delta_{ij}\delta_{pq}$$

(the first term has been symmetrized, since the derivative of a symmetric tensor with respect to another symmetric tensor has an arbitrary skew part). For 3D problems, and using the usual ABAQUS conventions for the stress and strain vectors, this can be written in matrix form as

$$[D]=G\begin{bmatrix} 2 & & & & & 0 \\ & 2 & & & & \\ & & 2 & & & \\ & & & 1 & & \\ & & & & 1 & \\ 0 & & & & & 1 \end{bmatrix}+K_b\begin{bmatrix} 1 & 1 & 1 & & & 0 \\ 1 & 1 & 1 & & & \\ 1 & 1 & 1 & & & \\ & & & 0 & & \\ & & & & 0 & \\ 0 & & & & & 0 \end{bmatrix}$$

$$K_b=\frac{p_t}{3}\frac{(1+e_0)}{\kappa}\exp\left\{-\frac{(1+e_0)}{\kappa}\varepsilon_{kk}\right\}-\frac{2G}{3}$$

2.2 Implement this procedure as an ABAQUS UMAT subroutine in EN234FEA. You can find relevant documentation on the course website: briefly, the subroutine must calculate the 'true' stress vector (stored using the appropriate ABAQUS conventions for 2D or 3D), and the material tangent stiffness **D**. For small strain problems, you can do the calculation using the strain components provided in the STRAN vector (it is best not to use this strain measure for large deformation problems, because represents an approximate strain measure that does not usually appear in finite strain constitutive equations).

EN234FEA contains an input file called Abaqus_umat_porous_elastic.in that you can use to test your user material subroutine. EN234FEA includes a procedure to check that the stress calculated in a UMAT is consistent with the material tangent matrix (by calculating a numerical derivative of the stress vector with respect to the strain vector and comparing the result to the DDSDDE matrix). The input file as provided has a STOP statement just after the CHECK MATERIAL TANGENT key – once the tangents are consistent you can remove this and run the analysis.

2.3 Once your UMAT works in EN234FEA, run it in ABAQUS/CAE. Set up a problem with a single 3D rectangular element, that is subjected to displacement boundary conditions representing a displacement controlled uniaxial tension test. Use ABAQUS to plot the predicted stress-strain curve for the material, and compare this with the EN234FEA solution.

As a solution to this problem (1) Please push your code for 2.2 to Github, and (2) include your solutions to 2.1, 2.3 in the pdf uploaded to Canvas.