PGE 383 - Advanced Geomechanics - Final Exam

Your final examination is to implement a plasticty return algorithm for the case of **isotropic linear hardening plasticity**. You can use the provided *Mathematica* code as a template to modify or otherwise as a resource for creating your own implementation.

Mathematica Template Code (.nb)

Mathematica Template Code (.pdf)

Use resonable material properties representative of *steel* and produce a plot of the *engineering stress as function of engineering strain* including separate curves for at least *two different hardening moduli*.

Submit a working version of your code to Canvas by **Wednesday, December 10, 12:00 noon**. You can upload any supplemental material or explainations to Canvas as well or otherwise hand deliver to me during the first hour of the scheduled final examination period, i.e. **9-10 AM, Wednesday, December 10**.

You are allowed any resource (e.g. books, notes, web) to complete the exam except consultation with another student or professor. You are reminded of the UT Student Honor Code, violations of academic integrity with be reported.

Further instructions are different for the undergraduate and graduate students and are detailed below.

Undergraduate Students

You only need to implement the plasticity algorithm for a *single element* undergoing uniaxial tension. If using the provided *Mathematica* code as a starting point, this should only require modification of the `computeStress[]` function and a change to the boundary condition application to apply uniaxial tension.

Graduate Students

Use the provided files detailed below which describe a mesh of a standard "dogbone" tensile test specimen commonly used in standard test frames. The mesh was constucted using QUAD9 elements to allow reuse of code from Homework Assignment 6, but feel free to reduce the degrees-of-freedom to QUAD4 if you wish, and use any integration scheme you feel is appropriate. Assume plane stress with thickness $t = 0.31\text{cm}$. The initial width of the gauge section is $W = 0.635\text{cm}$ and the overall length of the sample is initially $L = 10.16\text{cm}$. All coordinate distances are in centimeter.

coords.csv - the geometric node locations for all nodes. They are listed in x, y pairs with each line corresponding to a global node index starting with 1 and proceding in sequence.

connect.csv - the connectivity arrays. Each line contains the global node numbers of an element with local node numbering as specified in the schematic. For the pressue interpolates, only use the first 4 which will coorespond to the corners of the element.

nodeset1.csv - the nodes on the upper boundary. Use these nodes to specify a prescribed and/or fixed displacement.

nodeset2.csv - the nodes on the lower boundary. Use these nodes to specify prescribed and/or fixed displacement.

An image of the mesh is shown below for reference.

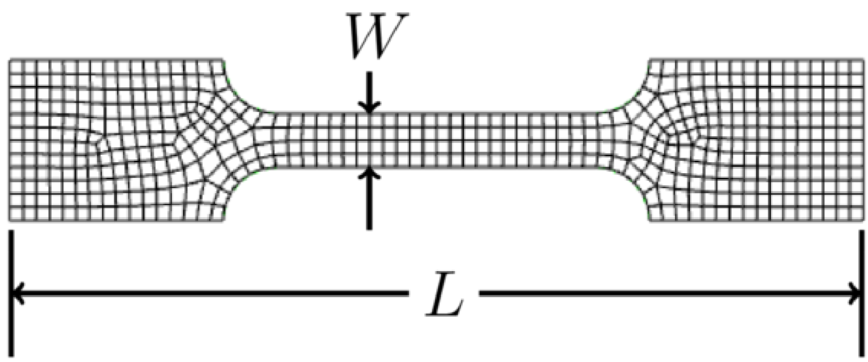

Figure 1: png