Tensile Test Specimen Deformation (15 points)

Objective:

1. Simulate nonlinear elastic deformation and plastic yielding.

A tensile test is a mechanical test in which a standard material specimen is subject to loading tension until failure. The resulting elongation and failure point inform engineers about the behavior of the material in structural applications. Since the results of any given test are compiled for standard usage by engineers, adherence to a strictly defined set of ASTM protocols is advisable.

You will create a specimen geometry in two dimensions which adheres to the specification given below. You will then carry out a two-dimensional simulation to yielding using `SHELL181` elements with thickness 0.254 mm and full integration (keyopt 3). Store all layer data (keyopt 8). Use `SECTIONS→SHELL→LAY-UP` to set the thickness.[*]

The specimen will be made of nitinol, a shape memory alloy which transitions between austenitic and martensitic phases. The test specimen is initially at 253.15 K and is loaded to 70 MPa to obtain a detwinned martensitic structure. The material is then unloaded to obtain a martensitic structure with residual strain, before finally being heated to 259.15 K to regain the austenitic structure.

Nitinol is a shape-memory alloy of nickel and titanium, with $\nu = 0.33$ and $\rho = 6.45\times10^{-6}$ kg·mm^{-3}. Shape-memory alloys have unusual cyclic characteristics and can restore their original shapes after plastic deformation [Kim2010]. The austenitic phase has $E = 70{,}000$ MPa and $\mu = 0.33$; the martensitic phase has $E_m = 70{,}000$ MPa. The phase transition is governed by the parameters:

$h = 500$ MPa $\quad R = 45$ MPa $\quad \beta = 7.5$ MPa·K^{-1}

$T_0 = 253.1$ K $\quad m = 0$

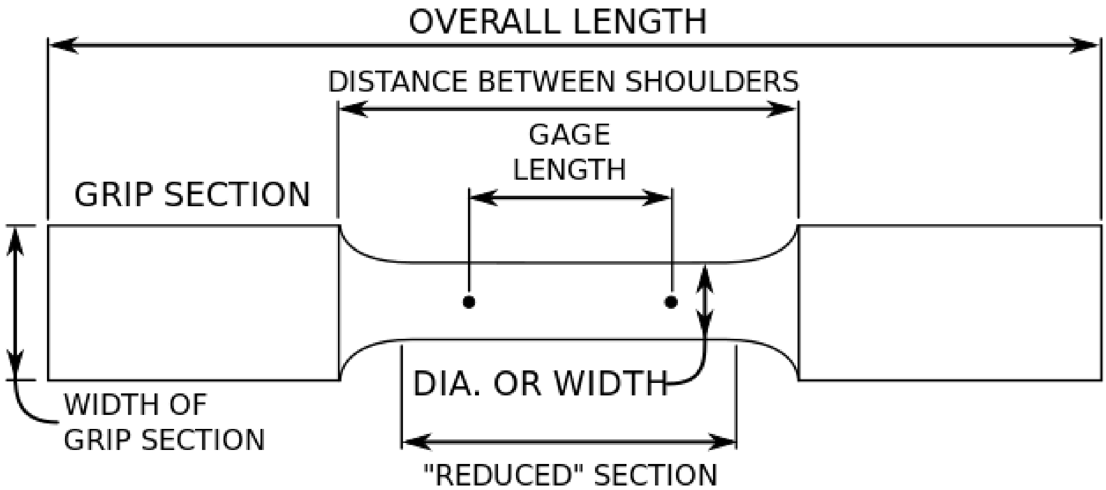
A script which solves a very similar problem using nitinol is available on the course web site as `sma.inp`. Run and examine this script to find out what it does, then modify it to solve this problem. (You do not, of course, have to use this script as the basis of your final solution, but it is a convenient reference point.)

Geometry

Since the physical object you are modeling is symmetric, as are the loading conditions, you can specify only the upper left-hand portion. Create a model with the following parameters.

Overall length	50 mm	Length of reduced section	15 mm
Width of grip section	12.5 mm	Width of reduced section	3.125 mm
		Reduced section narrowing radius	5 mm

*If `SHELL181` is misbehaving in your simulation (due to applying the pressure boundary condition, for instance), you may use a different element type (including moving to 3D) as long as you document your rationale and any changes made.



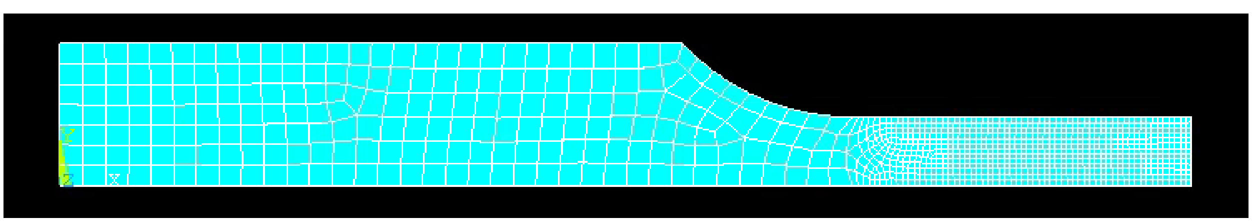
Representative coördinates for your geometry construction are:

- Rectangle (0, 0)×(25, 6.25)
- Circle (17.5, 9.375) r6.25
- Rectangle (17.5, 3.125)×(7.5, 3.125)

Your geometry model should be quite similar to the following figure.

Meshing

As you are using quadrilateral or hexahedral elements for the mesh, the Smart Size feature will not work. Thus you need to specify a free quadrilateral mesh subject to size controls along the lines. Along the neck, you should specify 0.125 mm as the element size. Other areas can use 0.25–0.5 mm, as they do not concentrate the force. (The details of this may be affected by how you have set up your geometry; simply try to get close to these settings.) A representative mesh is depicted below. You may need to divide the bottom line to facilitate smooth mesh transitions.



The right-hand side and lower boundary represent symmetry conditions. Set other loads as appropriate.

Doubly Insulated Pipe **(15 points)**

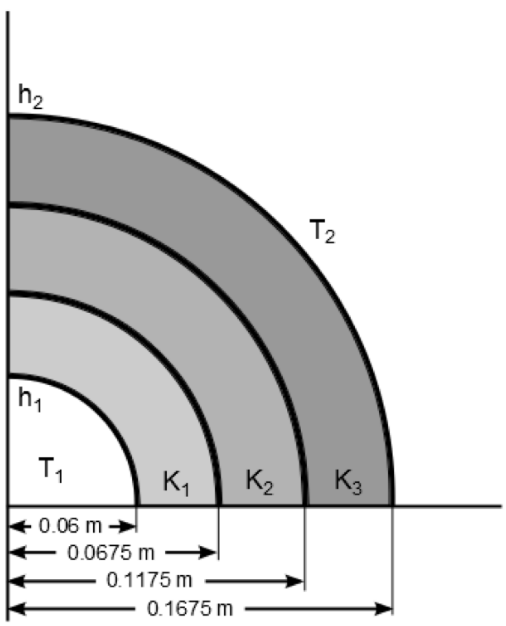
Objective:

1. Use scripting with parameterization.
2. Understand how to use thermal elements in APDL.

Consider a pipe system in which an inner pipe of ID 0.12 m carries hot steam at temperature T_{in} = 230 ℃. The pipe material has thermal conductivity k_1 = 49 W·m^{-1}·K^{-1} and an inner convective heat transfer coefficient h_1 = 85 W·m^{-2}·K^{-1}. Two layers of insulation surround this pipe. One is 0.05 m thick with thermal conductivity k_2 = 0.15 W·m^{-1}·K^{-1}, while the next is 0.05 m thick with thermal conductivity k_1 = 0.48 W·m^{-1}·K^{-1}. The exterior is exposed to air with a convection coefficient h_2 = 18 W·m^{-2}·K^{-1}.

Simulate this problem using 2D `PLANE55` and `PLANE77` elements and 3D `SOLID278` and `SOLID279` elements. You should do this programmatically, similar to the example given in class which used several element types in a single simulation (`cyclic.sh`). You may solve this case once to generate a base script which you can retrieve and clarify from the log file. Include your script with your submission. (You do not need to perform any coördinate system transformations to complete this exercise.)

Report the interface temperatures at radial locations x = 0.06 m, 0.0675 m, 0.1175 m, and 0.1675 m. Target temperatures are 222.3, 222.2, 77.040, 48.030 ℃, respectively.



Report

Document both simulations in a 7–10 page report (with figures) containing the sections:

- Problem description (specimen shape, grid, etc.)
- Numerical values (element parameters including any nonstandard keyopts, number of nodes, boundary conditions, etc.)
- Observations of numerical behavior
- Discussion of the physics

Include the following plots in your report as appropriate, with data from each case:

- Mesh of model
- Line plot of results along principal axes of note (*i.e.*, the temperature along a radius of the pipe or the stress along the centerline of the tensile test specimen)

The report should be formatted with 1.5 line spacing, 1 inch margins on all sides, and set in 11 point serif font. Figures and tables should be numbered with labels and captions.

Reference

Kim2010: K. Kim, K. A. Juggernauth, S. H. Daly. (2010) Stress-induced martensitic phase transformation in nitinol under hard cyclic loading . *Proceedings of the ASME 2010 Conference on Smart Materials, Adaptive Structures and Intelligent Systems (SMASIS2010).*