Homework #4 (HW4) Assigned 3/15/22
BIOEN 5250/6250, Fall 2022 Due online 3/29/22, 9 am MST

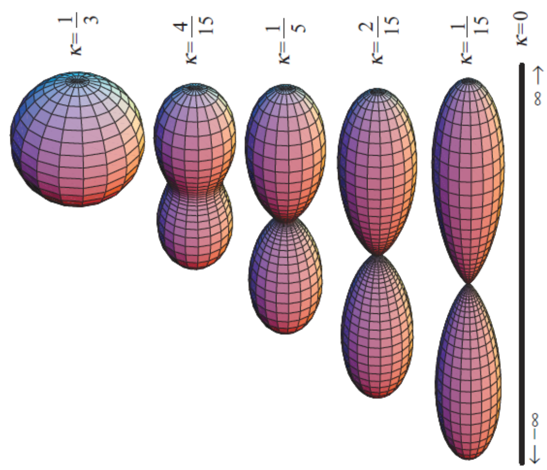
You must follow the homework guidelines on Canvas (Files/Homework/ BME_5250_Instructions_for_Preparing_Homework.pdf). **Submit any code and output along with your homework. When answers are algebraic expressions, you must either write the expression by hand, type the equation in a word file, or use the output of a livescript file in Matlab**. Please Submit completed homework on Gradescope before the beginning of class on the due date.

---

1. (Modeling fiber dispersion for transversely isotropic materials) Soft tissues such as skin and arteries often exhibit distributions of fiber families. For instance, in the manuscript by Annaidh *et al.* 2012, the authors used a π-periodic von Mises distribution to describe the distribution of fiber density in skin as a function of angular orientation. The Holzapfel-Gasser-Ogden (HGO) hyperelastic constitutive model was developed specifically to model fiber dispersion (Gasser et al., J. Royal Soc Interface, 2005, see the pdf in Canvas). In this constitutive model, the standard structure tensor $\boldsymbol{a}^0 \otimes \boldsymbol{a}^0$, is replaced with an alternative tensor $\boldsymbol{H}$ that is a function of a fiber dispersion parameter κ:

$$\boldsymbol{H} = \kappa \boldsymbol{1} + (1 - 3\kappa)\boldsymbol{a}^0 \otimes \boldsymbol{a}^0 .$$

When $\kappa = 0, \boldsymbol{H} = \boldsymbol{a}^0 \otimes \boldsymbol{a}^0$, which is the standard form for a transversely isotropic material with one family of fibers aligned along a single direction. When $\kappa = 1/3, \boldsymbol{H} = \boldsymbol{1}$, which yields an initially isotropic distribution of fibers. If the distribution is described by the π-periodic von Mises distribution, the angular distribution density can be visualized as shown:



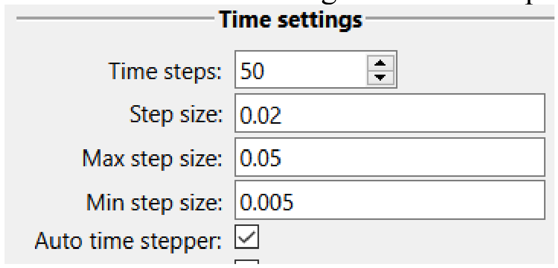
Construct a single-element finite element model with symmetry boundary conditions as you did in HW1, problem 5 to simulate the uniaxial elongation of skin along the *x*-direction to a stretch ratio of 1.5. Assign the HGO model to your element (Add Material → Uncoupled Elastic → Holzapfel-Gasser-Ogden). Use the following material coefficients and settings for your initial analysis, taken from Annaidh *et al.* 2012:

c	Shear modulus of ground matrix	0.2014 MPa
k_1	Fiber modulus	243.6 MPa
k_2	Fiber exponential coefficient	0.1327 (no units)
γ	Fiber mean orientation angle	0 degrees
κ	Fiber dispersion parameter	0 initially (no units)
k	Bulk modulus	100,000 MPa
Axes	Vector	
a	Primary fiber direction	1,0,0
d	Secondary fiber direction	0,1,0

Please see the FEBio manual page if you are interested in learning more about the HGO material:

https://help.febio.org/FebioUser/FEBio_um_3-4-4.1.2.6.html

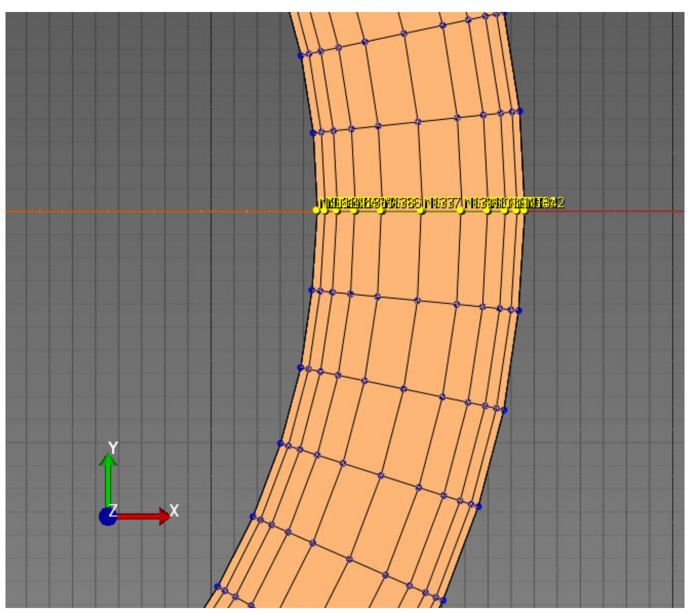
Add an analysis "Step" for structural mechanics and change these timestep settings:



Run your finite element model for the 6 values of the dispersion parameter κ shown in the figure above and create an x-y graph of the uniaxial Cauchy stress versus stretch ratio (1.0 to 1.5) containing all 6 stress-stretch curves. Briefly interpret your findings (1-2 sentences). Include a pdf of one of your .feb files with your homework upload for this problem.

2. (Residual Stress) The objective of this problem is to investigate the effect of pre-strain on circumferential stress in a finite element model of a carotid artery with ring opening angle of 90^{o}. You will use two FEBio Studio models named "Artery.fsm" and "Artery_Residual_Stress.fsm" that are available on Canvas. The first model has 1 analysis step consisting of inflation via a luminal pressure, while the second model has 2 steps, the first pre-straining the ring to close, and the second applying the luminal pressure. Each analysis step has a time span of 1. The luminal pressure is 20.0 kPa (approximately 150 mmHg), and is considered high blood pressure. The constitutive model is a homogeneous isotropic Veronda-Westmann material that is nearly incompressible.

   a. Open the model Artery.fsm. Take some time to inspect the boundary conditions and applied loading in the model tree so you are familiar with the model setup. Then run the model on your local computer. Inspect the circumferential stresses through the wall thickness. Go to the unloaded configuration, Time = 1 (the final analysis step). Select the row of nodes along the positive x-axis using shift+click+drag. Make sure FEBio Studio is not also selecting the nodes behind the front ones.



Create a graph of the circumferential and radial stresses as a function of radial position ("Post → New Graph"). Use the graph *Type* "Time-Scatter". Select X-Position for the *X*-axis. Since the edges of the elements connected to the nodes you selected are tangent to the y coordinate direction, the circumferential stress is equivalent to the Y-Stress at those locations. So select Y-Stress for the y-axis.. Then hit *Tools*, and choose *Current time step*, so you only see data for the current state. Attach a screenshot of the graph, and describe the state of circumferential stress through the wall thickness. Do the same for the radial stress (X-Stress). Export the data for the circumferential stress as you will need it in the next part of the problem. Briefly describe the distributions of radial and circumferential stress, do they make sense to you based on the loading?

b. Now open the model "Artery_Residual_Stress.fsm". Again inspect the boundary conditions and applied loading in the model. Also inspect the two Steps in the "Step" section of the model tree. If you expand the items in each Step, you will see the different boundary conditions for closing the artery (time 0 to 1) versus inflating the artery (time 1 to 2). Then run the model on your local computer. Open the results plotfile and navigate to Attach a screenshot of the same circumferential stress graph at Time = 1, and describe the circumferential stress throughout the vascular wall after application of prestrain.

c. Now go to the loaded configuration, Time = 2 in the model "Artery_Residual_Stress.fsm", and plot the wall thickness position versus circumferential stress as you have done previously. Export the data for the circumferential stress. Using these data and the data from part a, create a plot containing the circumferential stress as a function of radial position through the arterial wall for both models in a graphing software of your choice. Label the graph, include a legend, and make sure to include units. Then, describe how pre-strain affected the distribution of circumferential stress through the thickness of the arterial wall in comparison to the model with no pre-strain.