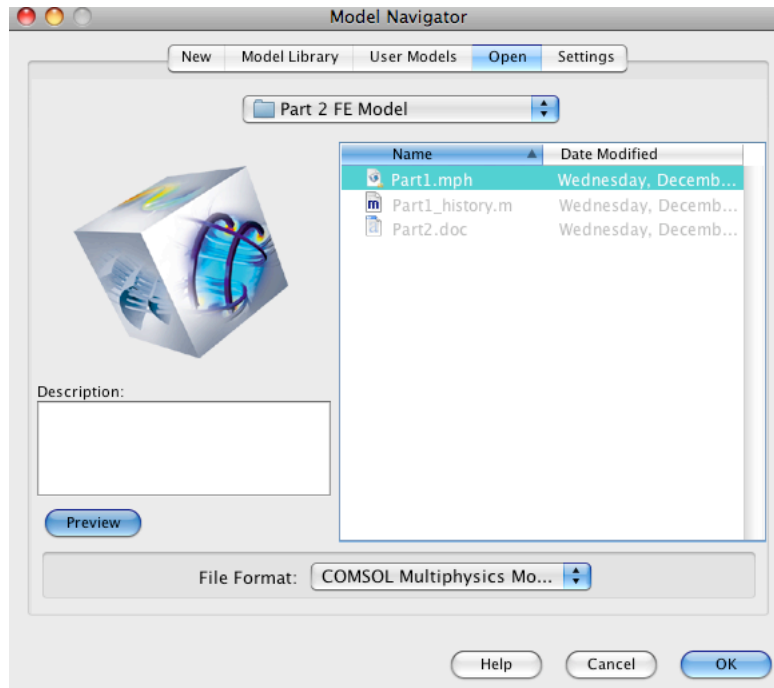


Analysis of a Hip Replacement: Part 2 – Finite Element Modeling and Analysis

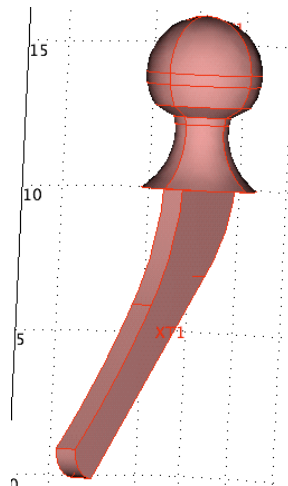
The Ohio State University: BME 643: FEM in BME
Autumn 2009
R.T. Hart (hart.322@osu.edu)

3.5.a1 6/2/2009

1. Start the COMSOL application as before, use the pull-down menu on “Open” and select the appropriate model, and then hit OK:



This should open the model in the state in which it was last saved in Part 1.

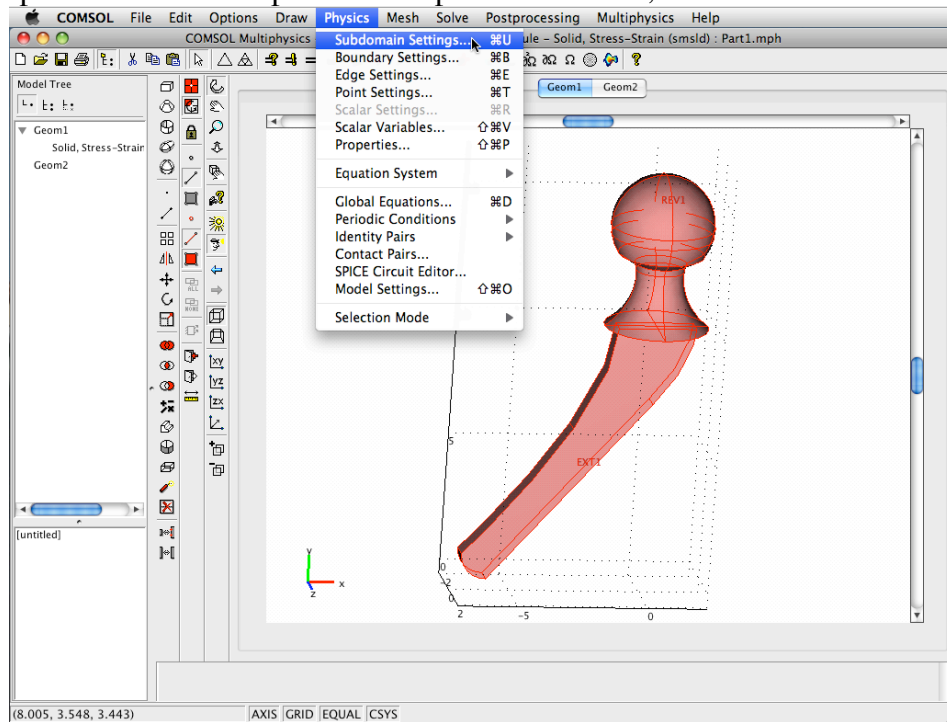


2. We are going add information to the 3-D geometry on our path towards a finite element model that can be analyzed. The fundamental branch of physics we will use for the analysis is **mechanics**. (Using the finite element method and the COMSOL program, we could also do a thermal analysis, or an electrical analysis, or...). For this problem in order to convert the 3-D

geometry into a model that can be analyzed to examine its mechanical behavior, there are several steps that we will need to perform:

- Material property assignment for the model (the **subdomain**). What is this made of, and what are the relevant material properties of the constituent materials?
- Assignment of **Boundary Conditions**. The model does not float in space – it is held in place by the surrounding bone. What are the prescriptions of displacements that we need to apply to simulate the surrounding bone?
- Assign the **loading** force to the model. How does the body load the hip?

Material Properties: From the top PHYSICS pull-down menu, select the Subdomain Settings:



(Note: for a 3-D model, the **subdomain** is the volume, the **boundary** is a surface in space, an **edge** is a curve in space, a **point** is a location in space).

In this case, we have 2 subdomains (1 is the stem, 2 is the collar-head) and we will assign the same properties to both (although some femoral hip components use different materials for the head). Note that we have used dimensions of centimeters to measure length, but not made any settings to take that into account.

To set the units for an existing model, go to the **Physics** menu, choose **Model Settings**, and then select the **Base unit system** to be **CGSA**, then **OK**.

“CGSA units. The CGS system uses centimeter, gram, and second as basic units of length, mass, and time, respectively. The remaining basic units are identical to the SI units. The CGS unit system gives nice values for small lengths, masses, forces, pressures, and energies when working on a microscale and with weak electromagnetic forces. The derived units of force, pressure, and energy have well-known and widely used names: dyne, barye, and erg, respectively. CGSA adds *ampere* as the basic unit for electric current.”

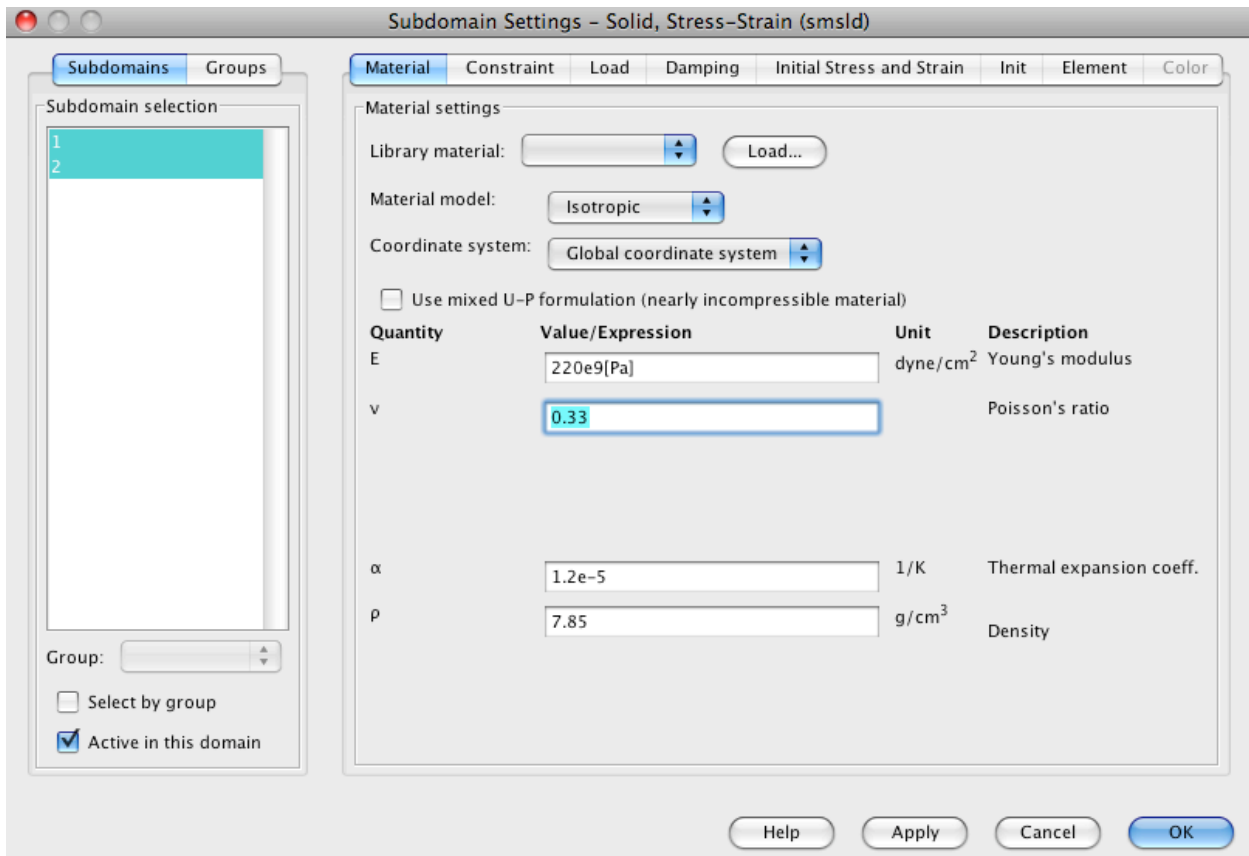
The modulus, measured in Force/Area should in baryes (dynes/centimeters²), requiring a conversion from tables that list properties as Pascals, which are Newtons/meters². One bayre is 0.1 Pa.

A metal that has FDA approval for orthopaedic implants is an alloy made of cobalt chromium and molybdenum. A reasonable value for the Young's modulus is 220 GPa, and 0.33 for Poisson's ratio. So for this table we will use a modulus value of:

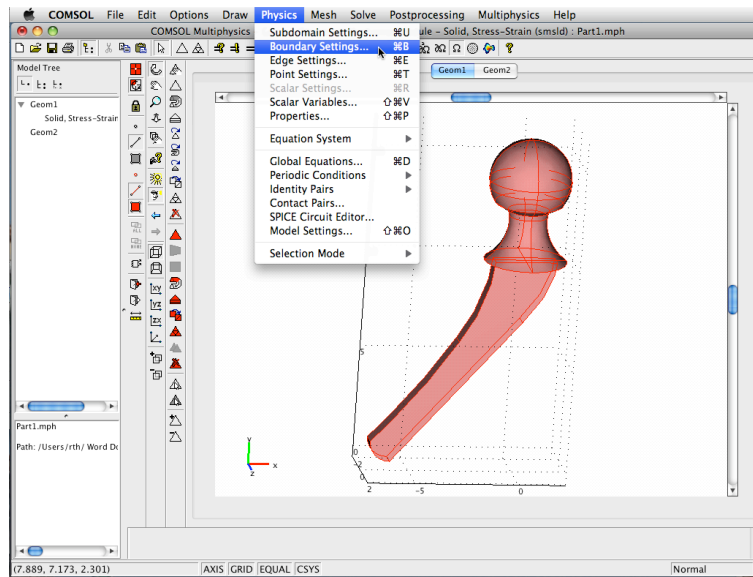
$$220 \text{ GPa} = 220 \times 10^9 \frac{\text{N}}{\text{m}^2} \cdot 0.1 = 228 \times 10^8 \frac{\text{dynes}}{\text{cm}^2} = 2.20 \times 10^{10} \text{ baryes}$$

We could either use this value, or (newly available in a recent COMSOL update), use the value with the units we want (labeled with square brackets, [Pa]), and let the program take care of the conversion.

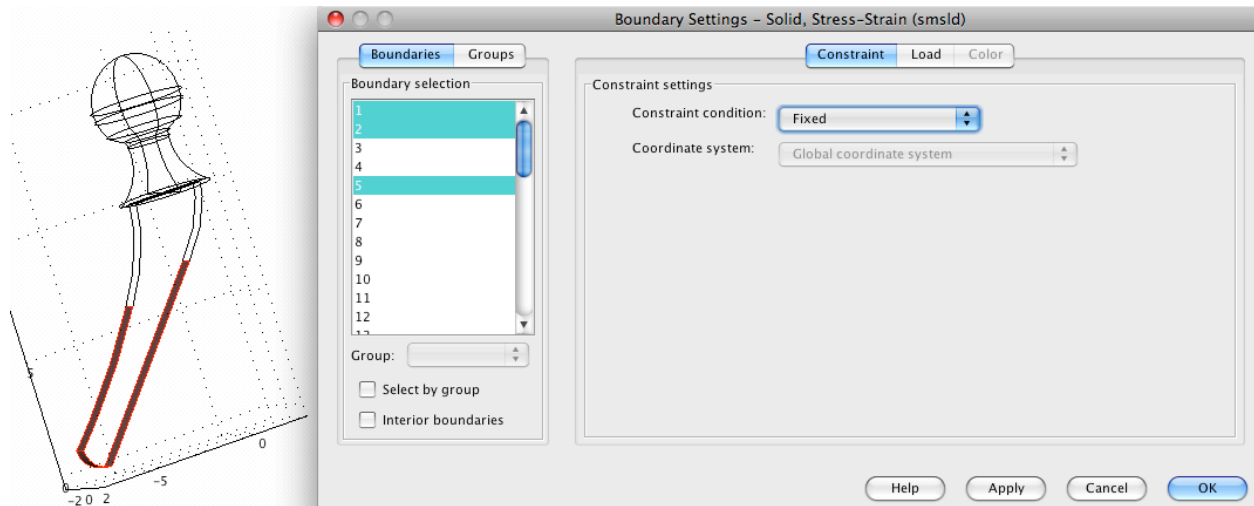
The other properties (density, thermal expansion, etc.) are not going to be used for the static analysis we will be performing, so we'll leave them at the default values.



Constraints: From the PHYSICS pull-down menu, now select Boundary Settings:



Although in reality, the femoral hip component would be well supported by the surrounding bone, we're going to apply boundary conditions that *idealize a drastically loosened stem* (that would require revision surgery). (This will make it easier to apply the boundary conditions, and also lead to **stress distributions that are more interesting**.) So we will choose just a few of the surfaces (specifically 1,2, and 5) to constrain (prevent motion in the x-, y-, and z-directions):



To get the highlighted surfaces, you may either click on the model or on the appropriate surface number. To get multiple surfaces, hold the 'ctrl key' while selecting (on Macs hold the 'Apple' key on Macs). Set the Constraint Condition to "Fixed" for these surfaces only and hit APPLY.

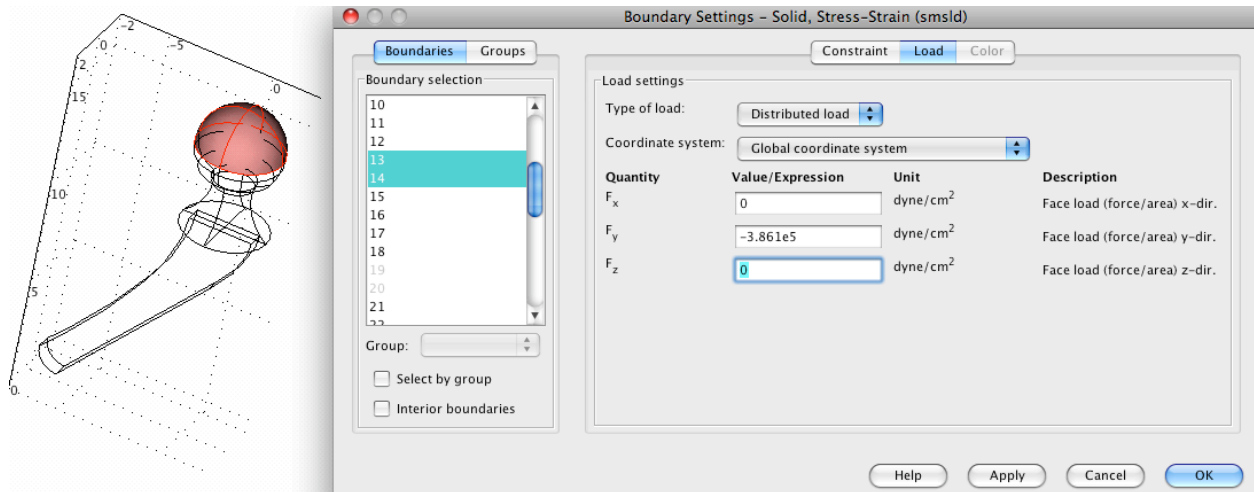
From the top of the same pull-down, we will apply forces to the top of the femoral head. First, click LOAD on the top of the window (so that it is highlighted instead of "Constraint").

We will put a load in the $-y$ -direction, just on the top surface of the femoral head. For this model, that turns out to be 4 surfaces (boundaries), 13,14 and 38 and 39. The load should be measured as a pressure (force/area) and just like for the modulus we will use Pascals =

Newtons/meter² (1 Newton is approximately equal to the gravitational force generated by the mass of an apple).

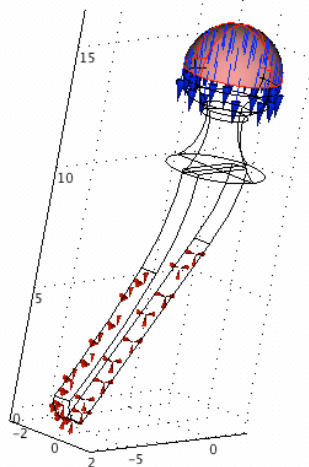
In extreme cases (e.g. climbing steps), the force on the hip may be 3.5 times body weight. Assume body weight is 160 lbs and assume that this is distributed over 1 in². The relevant conversion factor we want is to multiply lbs/in² by 6894.8 to convert to Pascals. So:

$$3.5 \left(160 \frac{\text{lb}}{\text{in}^2} \right) = 560 \frac{\text{lb}}{\text{in}^2} \cdot 6894.8 = 3.861 \times 10^6 \frac{\text{N}}{\text{m}^2} \cdot 0.1 = 3.861 \times 10^5 \text{ baryes}$$



(Note this is in the $-y$ -direction, i.e., downward.)

If you want to check to make sure the boundary conditions and loading has been correctly applied, go to the top pull-down OPTIONS menu and select 'Show symbols' to get a plot showing regions that are loaded and regions that are constrained:

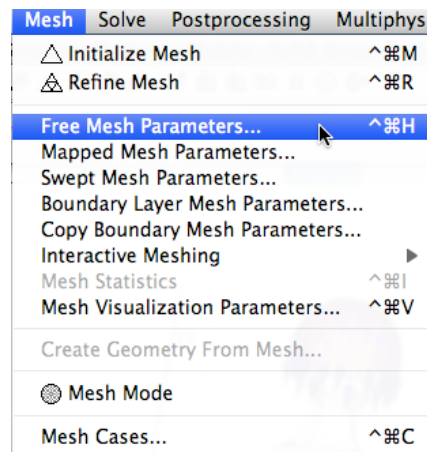


3. The next step is to subdivide the geometry into a finite element mesh – a collection of small, regularly shaped, geometric elements. The finite element method is based on calculating the mechanical behavior (in this case the stiffness, which is the resistance to deformation) for each of the small elements (involving numerical integration over the volume of each element) and then

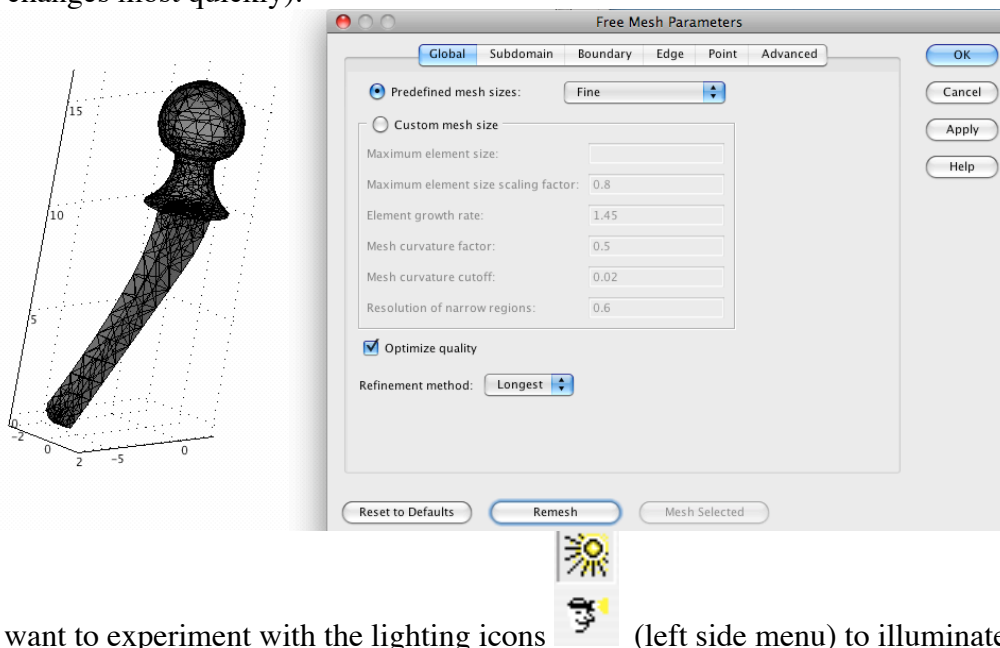
summing the stiffness contribution of each element into a large set of algebraic equations representing the stiffness of the entire (global) structure.

The process of meshing in 3-D used to be quite challenging. However, the COMSOL software will allow us to generate excellent quality (quadratic) tetrahedral elements almost instantly. One challenge that remains is to balance accuracy (in general, we get better answers with more elements) with the needed computational resources (fewer elements for faster processing). For these analyses, we will use a relatively coarse mesh.

From the top MESH pull-down menu, select Free Mesh Parameters:

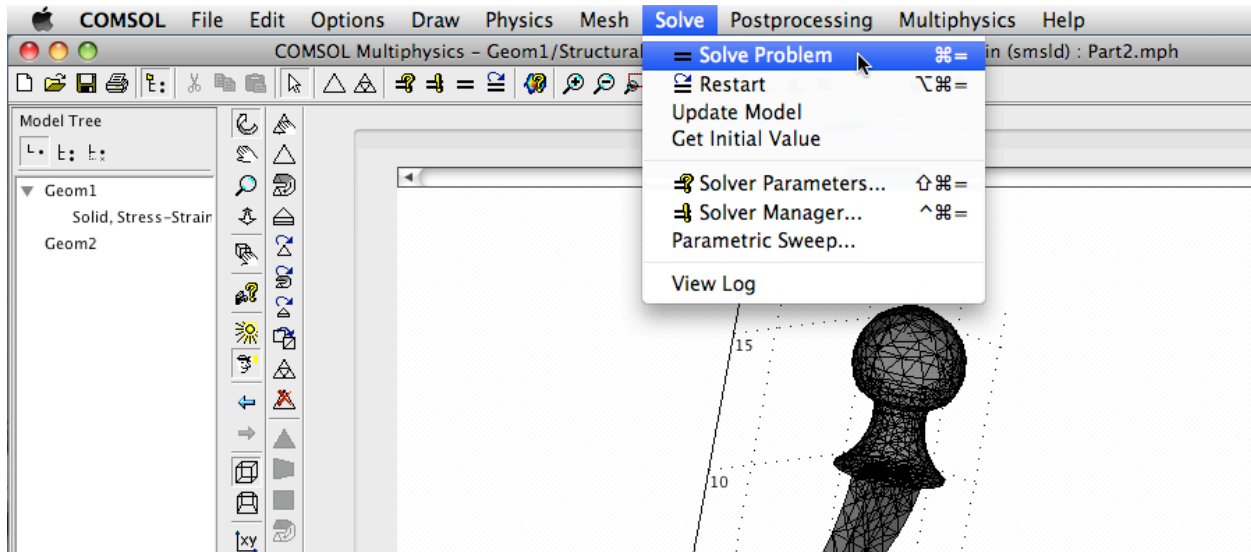


From the mesh parameters, select the “Finer” predefined mesh size, hit REMESH, and after a bit of processing, approximately 4000 elements (each a 3-D tetrahedron with quadratic sides) are generated to approximate the 3-D geometry (with more elements packed in locations where the geometry changes most quickly):



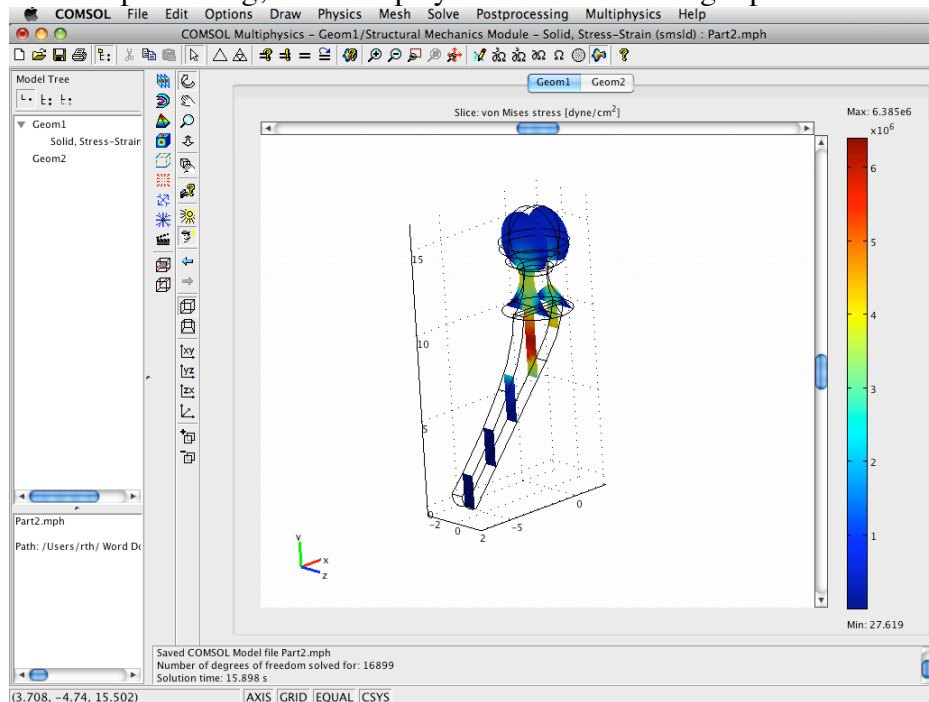
You may want to experiment with the lighting icons (left side menu) to illuminate the model. **Save your model, again.**

4. Although COMSOL has a variety of solver options, we will simply choose the default by selecting the 'equals sign' from the SOLVE pull-down menu:



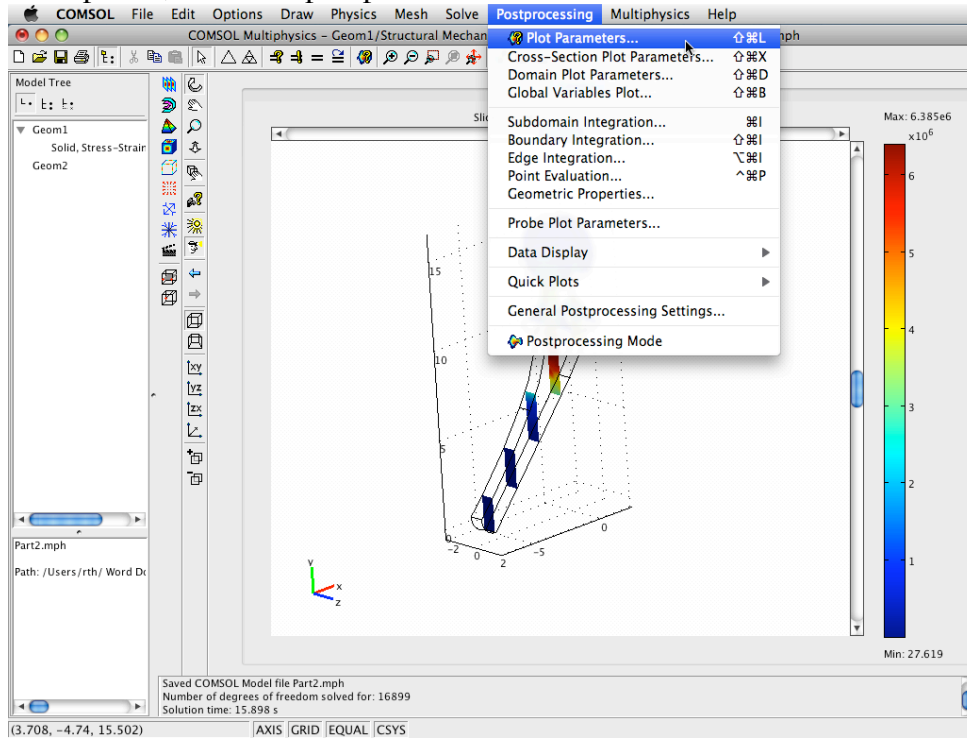
The program is generating a set of algebraic equations found by integrating over the volume of each of the finite elements (functions that include geometry and material property information). This information is assembled into the large set of simultaneous algebraic equations. For this model, the program solves a set of almost 20,000 equations in 20,000 unknowns. (Just 15 years ago, this would have required a supercomputer and hours of processing time). This is solved on my laptop in just under 5 seconds, although a progress bar is displayed to keep you informed and patient.

When it has finished processing, it will display a rather odd looking representation of the results:

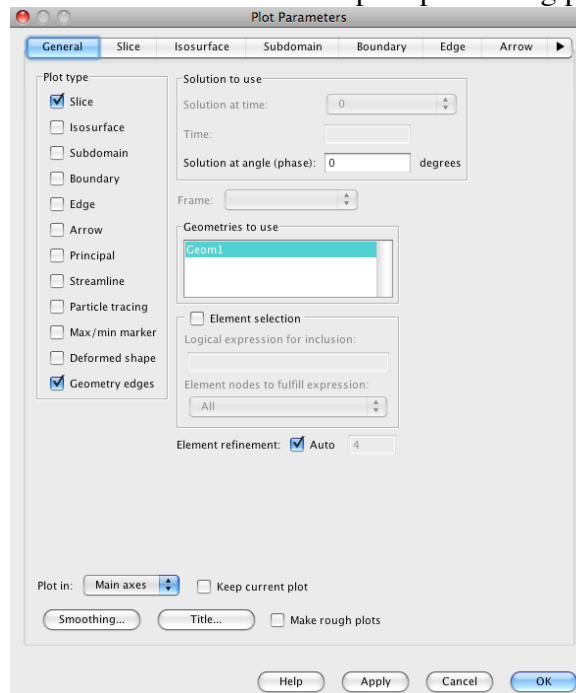


This figure shows a color ‘fringe plot’ of a composite stress measure (von Mises equivalent stress) presented on a series of 2-D slices through the model. This kind of plot allows the bioengineer to peer inside the structure and is just one example of the kind of ‘post-processing’ information now available to the analyst/designer.

To get to other options, select the plot parameters from the POST PROCESSING menu:

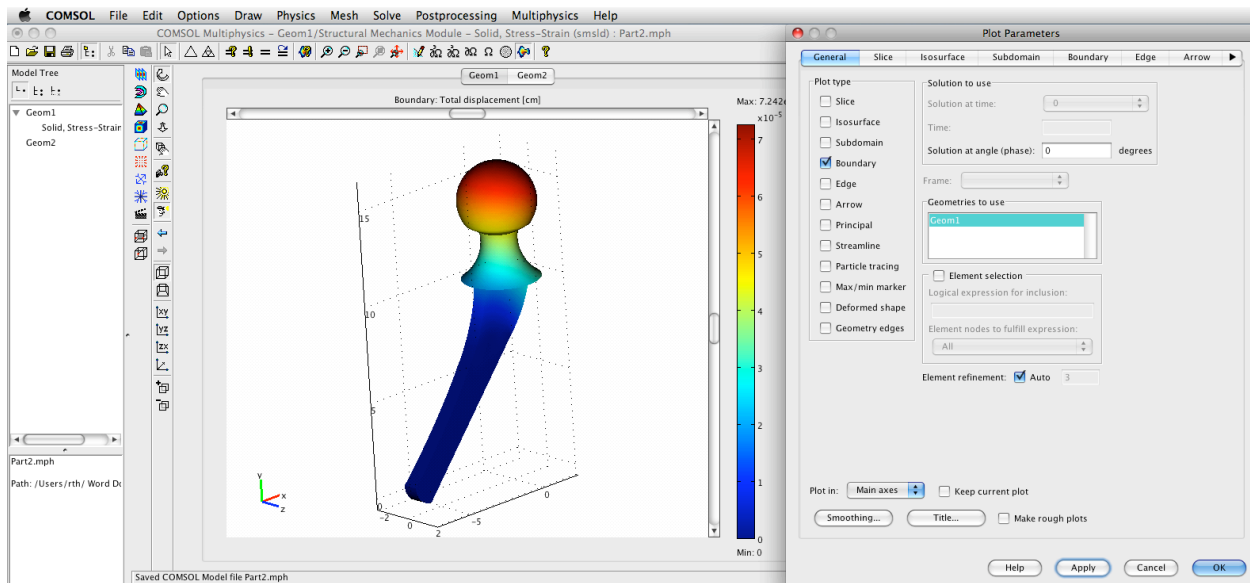


The resulting window will present a number of nested menus (shown across the top of the window) for a wide variety of options (so many options that an arrow is shown at the top pointing to additional parameters that don't fit in the post-processing plot options window).



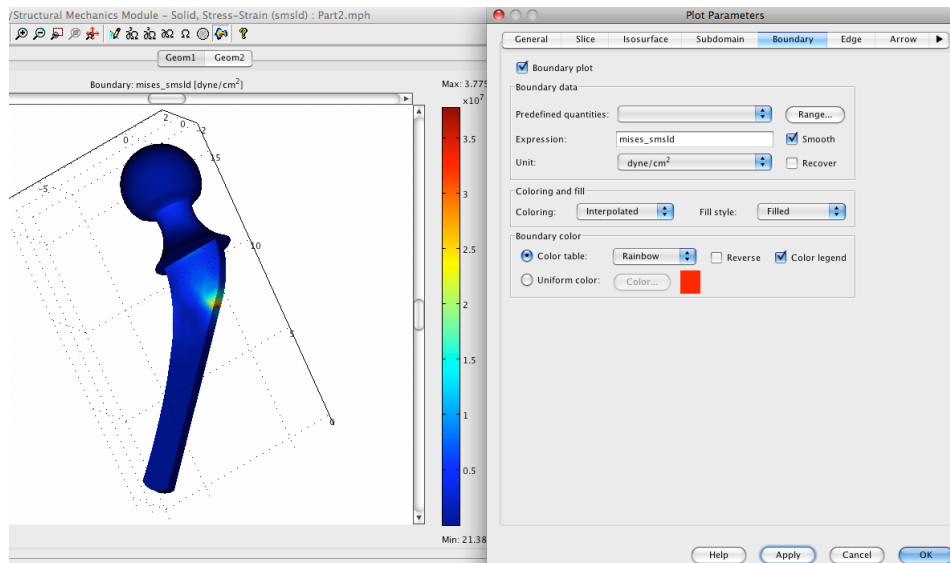
The screen that is showing, GENERAL, is indicated by the highlighted tab and summarizes the plot currently displayed (slice showing geometry edges. (If we were to select the SLICE tab, it would show that the component being displayed is the von Mises equivalent stress -- a positive scalar quantity calculated with a contribution from each of the 6 independent stresses – you'll learn about this in Mechanics of Materials).

To see a color fringe plot of displacement (should be large near surface of load application, should be zero at surfaces that were constrained by application of boundary conditions), toggle off the 'Slice' and 'Geometry edges' buttons, toggle on the 'Boundary' button, and hit APPLY:

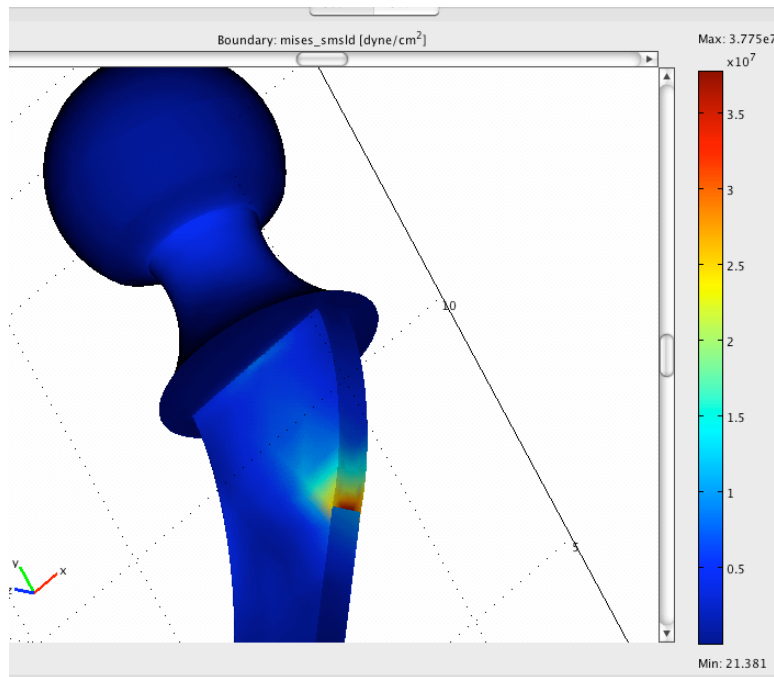


To see the von Mises equivalent stress on the boundary (a stress measure that is useful for predicting the location of potential failure in metals), we can simply change the data being plotted on the boundary.

Change the expression in the boundary window from 'disp_smsld' to be 'mises_smsld' and hit APPLY:

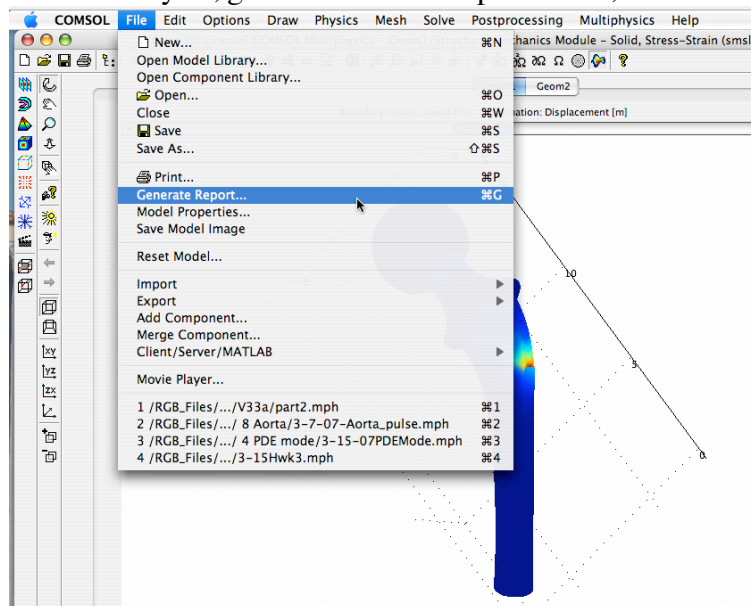


This shows a high von Mises stress concentration at the back of the stem that can be better seen from different views. Use the mouse to select a different view, such as:



Recall that our boundary conditions were an approximation that are for an extreme case with very little contact with the bone. But in this extreme case, and depending on the number of loading cycles, the specific materials, and the magnitude of the stress concentration we have seen, this is the place where we would predict initial failure of the metal stem. **YIKES! Save!**

To generate a report of the analysis, go to the main File pull-down, and select Generate report:



and select the “Generate” tab (with the options set at defaults, which is for a full report). This generates an html file, and a directory with the figures. Generate a pdf of the report by selecting print and then save to PDF.