BME 2000 Project Tips

General Advice:

- Divide the work up amongst you: while we want everyone to have experience with the ANSYS portion as well as the analytical, I would highly recommend assigning a couple people to each task, and then reviewing everything as a group when you write your paper and presentation.
- Even though in the homeworks we have usually compared the analytical portion directly to the numerical results by uploading a CSV, this isn't necessarily required for the project. Instead, compare trends like velocity profiles or concentration distributions, and think about how they should match up.
- As always, with any aspects of the project, email me with any questions you might have, and definitely come to office hours if you get the chance!
- Anyone working with diffusion should keep the heat-mass diffusion analog in mind. In order to properly use the Energy model in ANSYS to solve mass diffusiom problems, you'll need to change the thermal properties such that you get the diffusivity you're looking for. It was touched on in HW 3, but you'll need to use the following equation:

$$\alpha = \frac{k}{\rho C_p}$$

- Since density is an important property for the fluid flow equations, it needs to be accurate to the fluid. However, both thermal comductivity (k) and heat capacity (Cp) can be changed arbitrarily to achieve a certain Alpha. This alpha is analagous to solute diffusivity, i.e. the diffusion coefficient.

## Project 1: Shear Stress Bioreactor

Analytical:

- The first step here is to model the bioreactor as current controlled parallel circuit. Using the equation for fluid resistance in a rectangular channel and the known overall flow rate given by the paper. Here is the equation for resistance:

$$\text{Resistance:} \quad \frac{12\mu L}{\left(wh^3\left(1-0.63\frac{h}{w}\right)\right)}$$

- Once you have done this, you can find the flow rates through each individual channel, and use them to setup your ANSYS and the rest of your analytical steps
- Solve the navier stokes equation for flow between parallel plates under these conditions.
- Lastly, if there is time, try to derive the given equation for resistance in a rectangular channel using the navier stokes equation. Come to me for more help on this.

Numerical:

- For the ANSYS homeworks, we've been using relatively simple geometries. Here, we'll need to do a little bit more. I would highly recommend doing each individual channel, as opposed to trying to model the entire device.
- Using the given dimensions from the paper, start by sketching out the geometry from the top down for each of the three parallel channels. For the left and right ones, you don't need to include the horizontal inlet and outlet portions, just the channels that are parallel with the plate. Here is what each one should kind of look like, but with different dimensions of course:
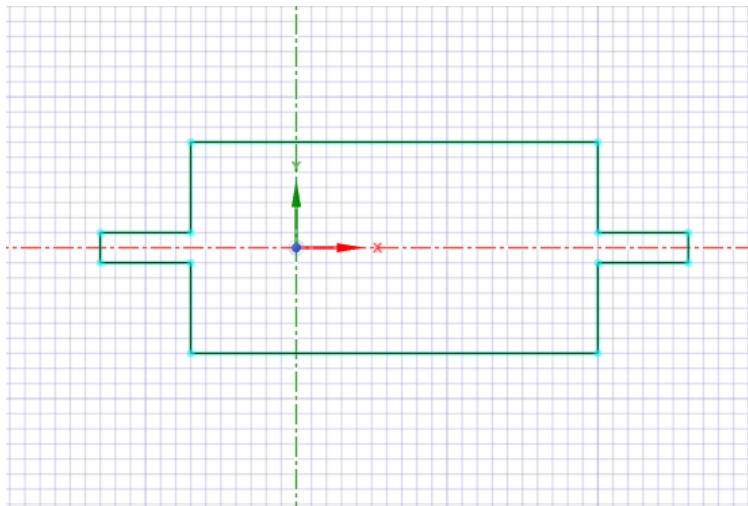
- After sketching on paper, your best friend is going to be the line tool. In spaceclaim, select the x-y plane and start a sketch. Using the line tool, draw lines following the paper's dimensions from the top down.
- Once complete, hit End Sketch editing and select the Pull tool. Pull the surface upwards by a length equal to the height of the channel. Voila! You have your geometry
- Lastly, for the meshing, remember to be very careful about your element sizes. My rule of thumb is to have at least 8-10 divisions between any two boundaries (like between the top and bottom of the channel). Using what you've learned for the homework, create the mesh so that this is satisfied. Come to me with any questions on this.

Project 2: Gliadel Wafer

Analytical:

- This portion is going to be extremely similar to the ANSYS HW 3, in that you have a concentration contained in an area which releases over time. Using Fick's 2nd Law, adapt the glioblastoma problem from the material4class.pdf handout from canvas to fit your particular situation. For the analytical, you should only be considering an axisymmetric spherical system, where solute diffuses from the center.

Numerical:

- Start with an axisymmetric model, very similar to what you did in the homework. Make ant changes to the system that you would need to do based on your own project.
- While we do want to have a *more* realistic geometry compared to the analytical, there's no need to go too crazy here. The aspects of the brain that we care about are asymmetries in its shape, and perhaps the perivascular spaces within the brain itself. To do this simply, I have a few tips:
  - Create a sphere using the sphere tool
  - Add a plane which bisects the sphere, and using the Move tool to put it around 1/3 of the overall height of the sphere.
  - Use the Split Body tool to cut the sphere using the plane, and delete the smaller portion. Now you have an asymmetrical brain that looks something like this:
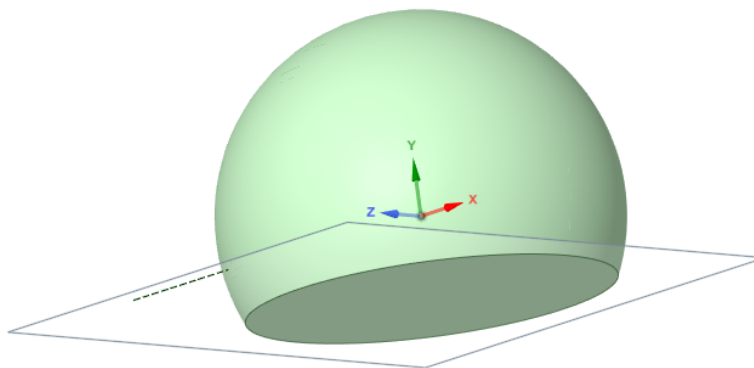
- Now you'll need to add the drug capsule. This can be done by making a small ellipsoid or sphere outside the brain, moving it inside using the Move tool, then cutting it out using the Combine tool. You'll see that a new solid has been added to the structure tree which you will want to delete.
- What you will have done here is sectioned out a part of the brain where the drug will sit, without making any gaps or overlaps.
- If you want, you can "carve" some vesicles out of the brain by doing the same process, but deleting both the created solid and the solid you used to make the cut. This can take some trial and error, and be sure to come to OH if you need any help.
- Here is an example of how that could end up looking, where I'm looking at a cross-section so you can see the hole
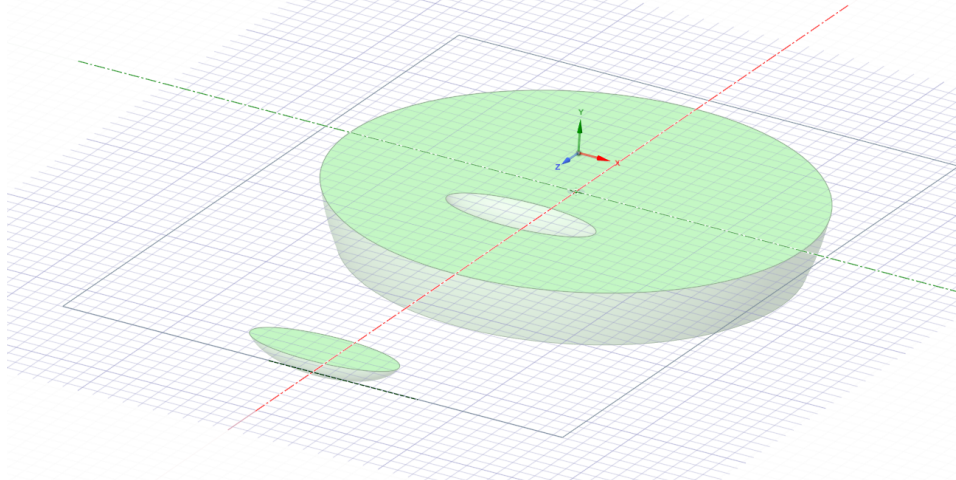
-
The rest of the numerical portion should be very similar to HW 3

## Project 3: Hemodialysis

Analytical:

- (Two parts: double parallel flow, Diffusion in three regions using B.C. and explain R = -k*C)
- Hemodialysis is one of the trickier things to model analytically, considering there is both solute diffusion and bulk flow coming into play. While our friends Truskey, Yuan, and Katz (the writers of the textbook for this class) have a very complex method of doing both at the same time, we're going to instead model each process independently.
- The first part will be the flow. You will need to model flow both inside and outside this hollow fiber. Since it's cylindrical, you have already done the inside when we covered Poiseuille flow. To model the outisde, see the Fall 2020 Problem Set 13 solution on canvas, or Fall 2020 Quiz 5 solution about flow around a catheter.
- The second part will be modeling the diffusion. Like the Numerical part, you'll need to define different solutions to the diffusion equation for the inner, outer, and membrane portions of the system. Using the cylindrical diffusion equation, define the appropriate boundary conditions for each section.
- For the outside region, you'll want to include a destruction term R. R can be defined as $-k*C(r)$, where k is a destruction constant. To save you all some work, you won't need to find this constant, and you can leave your solutions/relations in terms of some k when you present/report this.

Numerical:

- Using the rectangle tool, create three stacked rectangles representing the inner, membrane, and outer regions. While your literature will likely have an extremely thin membrane relative to the other spaces, for simulation reasons, you should keep the membrane around 2mm wide, or approximately 1/5 the height of the inner portion
- Here's an example of what that geometry might look like:
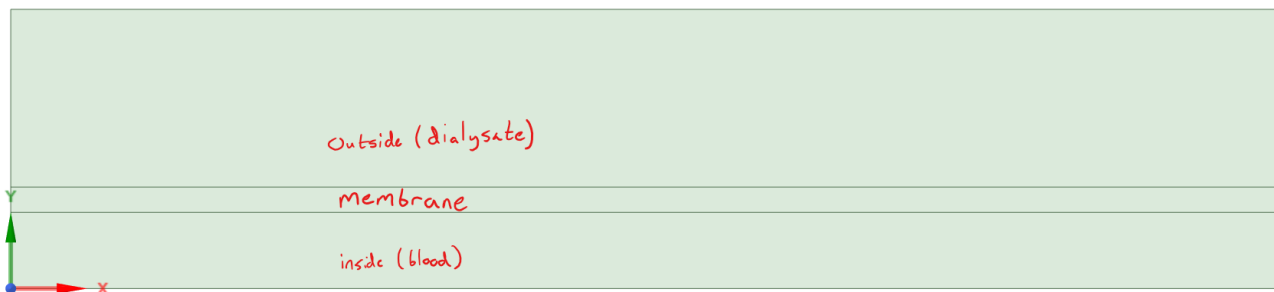
- Once you finish sketching, make sure to change your share topology by clicking on the design title (Design1 or FFF or something), and changing share topology to group under properties
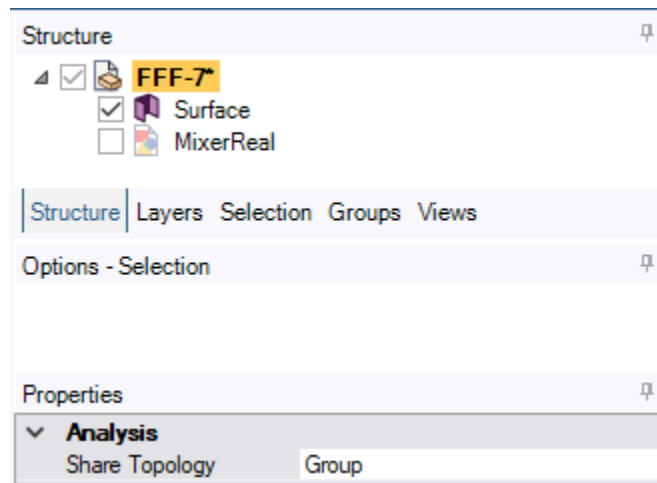
- As always, in the meshing, make sure that each layer has enough elements between boundaries, particularly the membrane here. At least 8 or 10 would be a good start.
- For the setup, you'll need to enable the energy equation under models.
- In materials, you'll need to make a solid material for the membrane as well as your fluid material. Make a new material, and change its type to solid. Just add the diffusive properties that should fit the membrane according to your literature. Reach out if you're having trouble here. Make sure to change the membrane cell-zone to this solid under cell-zone conditions
- Similar to HW 3, you'll need to take advantage of the heat/mass diffusion analog, and change the material properties such that you get the diffusivity you're looking for.
- Lastly, under your boundary conditions, you'll need to change one of your inlet temperatures (the blood) to 1 degree C, while the other should be zero. That should look something like this:
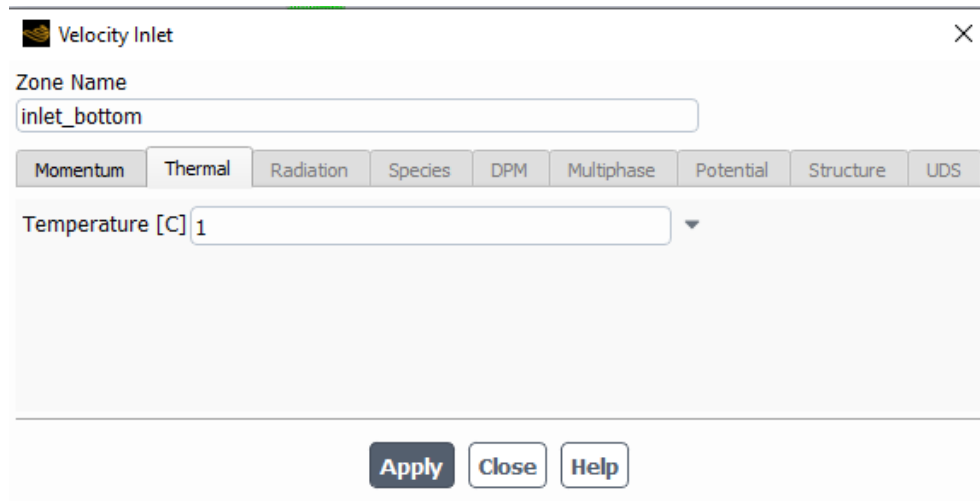
That's all I have for now, best of luck everyone! Please reach out if you need anything!