Fluid motion and mass transfer

I. VIDEO LECTURES

This week you should watch

- Videos for chapters 6 & 7 (Momentum, Stress Tensor, Newtonian Fluid)

II. READING

This week you should read

- Chapter 6 & 7

III. READING QUESTIONS

1. Explain, in words what the material derivative represents.
2. Why do we use a tensor to represent the state of stress of a material?
3. What is the physical meaning of the divergence of the stress tensor?
4. What is the differential form of conservation of mass for an incompressible flow?
5. When is the assumption of incompressible flow a good one?
6. What role does gravity play in a constant density flow?
7. What is the relationship between the stress tensor at a point and the stress vector acting on a surface passing through that point?
8. What's the difference between Euler's equations and the Navier-Stokes equations?
9. Physically, what is the reason that the stress tensor must be symmetric?
10. Physically, conceptually, and briefly, describe in words what is the constitutive law for a Newtonian fluid.
11. What is the boundary condition on the fluid velocity at a solid wall (state in words or mathematically)?
12. What is vorticity and what does it represent?
13. What are the units of the stress tensor?

IV. REYNOLDS NUMBER ESTIMATES

Estimate (order of magnitude) the Reynolds number for

- A car traveling at 55 mph.
- A hurricane (with 80 mph winds over 100 miles)
- A bacteria in water (2 μm in size moving at 100 μm/s)

V. COMSOL

1. Flow over a cylinder inside a channel.
    - Select a new model with “fluid flow”, “single phase”, “laminar flow”.
    - Select stationary
    - Create a channel that is 4 units high by 20 units long. Set the lower corner of the channel to be at $y = -2$ so the extent of the channel goes from $-2 < y < 2$.
    - Create a circle of radius 0.5 (diameter 1). Center it at $x = 5$, $y = 0$.
    - Use the Boolean operations to subtract the circle from the rectangle.
    - Under laminar flow set the fluid properties to be user defined, $\rho = 1$, $\mu = 1$.
    - The default boundary condition is no-slip.
    - Add an inlet boundary condition to the left wall at $x = 0$. Set the velocity inlet such that the x-component is $u(y) = (4 - y^2)/4$
    - Add an outlet boundary condition to the right most wall at $x = 20$.
    - Mesh and compute the solution.
    - Plot some streamlines - I like uniform density with a spacing of around 0.01.
    - Increase the Reynolds number by lowering the viscosity to 0.1. Re-run.
    - Increase the Reynolds number by lowering the viscosity to 0.01. Re-run.
    - If you continue to increase the Reynolds number, at some point the solver will die because physically the problem goes to one that is inherently unsteady. Exactly how this transition is predicted will depend on how fine your mesh is and other details of the solver that are beyond our course at this point. Set the Reynolds number to 500. Under Study Steps, disable the “stationary” and add “time dependent”. Solve for 100 units of time, outputting every 0.25. Re-solve and watch the animation.
    - Experiment with some of the different plots.
    - Write a short paragraph explaining your observations. Include one picture from Comsol that you think looks cool.
    - Save the model for the next problem.
2. Flow over a cylinder in free stream.
    - Edit the previous model to set the upper and lower walls at $y = 2$ and $y = -2$ to also be “outlets”.
    - Set the inlet velocity field to be $u = 1$ at $x = 0$.
    - Set the Reynolds number to 9.6.
    - Enable the stationary solver and disable the time dependent one.
    - Compare the computed flow field to the experimental images at the exact Reynolds numbers. Note, you don’t need to do a one to one match of the flow field but you should be able to easily measure the size of the seperation bubble behind the cylinder (relative to the diameter) and compare to your simulation results. You can be semi-quantitative here.
    - Those images claim in Figure 46 that the flow goes unsteady at $Re > 41$. While such accuracy will be hard for you to get, try switching to the unsteady solver and see about where you observe a transition from steady to unsteady.
    - Try the Re=2000, though you could have trouble getting it to solve. Don’t bother with Re=10,000.
    - Turn in a panel of images at each Re in the experimental images. Show streamlines and zoom in such that the image has approximately the same extents as the experimental images. Note the Re where you found the flow go unsteady.
3. See the images for creeping flow in a rectangular cavity. Reproduce the experimental results. Include a panel of 4 images to match the experiments.