Fluid motion and mass transfer

I. READING

This week you should read and watch

- Chapter 6
- YouTube lectures on Leibniz, material derivative, material derivative and Leibniz, and mass transfer. Some of these were previously defined.

II. READING QUESTIONS AND CONCEPT QUESTION

1. Explain, in words what the material derivative represents.
2. Explain in words, what does the dimensionless Peclet number represents.
3. Under what conditions would the equations $\frac{\partial c}{\partial t} = \mathcal{D}\nabla^2 c$ and $\frac{Dc}{Dt} = \mathcal{D}\nabla^2 c$ be exactly the same?
4. What principle does the equation $\nabla \cdot \mathbf{v} = 0$ represent, physically? When is it not a valid equation?
5. Under what conditions does $\frac{\partial c}{\partial t} + \nabla \cdot (c\mathbf{v}) = \frac{Dc}{Dt}$.

III. COMSOL FLUID FLOW

1. Flow over a cylinder
   - Select a new model with "fluid flow", "single phase", "laminar flow".
   - Select stationary
   - Create a channel that is 8 units high by 20 units long. Set the lower corner of the channel to be at $y = -4$ so the extent of the channel goes from $-4 < y < 4$.
   - 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 $u(y) = 1$ and $v = 0$.
   - Add an outlet boundary condition to the right most wall at $x = 20$ as well as the upper and lower walls.
   - Mesh. You can experiment with different refinement. The higher the Re, the more refined the mesh will need to be. Extra fine works well for lower Reynolds numbers. Extremely fine might be needed for higher ones.
   - Click solve 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.
   - 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. For this comparison, focus on Reynolds numbers of 9.6, 13, 26, and 41.

- If you continue to increase the Reynolds, 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 200. Under Study Steps, disable the “stationary” and add “time dependent”. Solve for 100 units of time, outputting every 0.5. Re-solve and watch the animation. You should clearly see the flow becoming unsteady at this point.
- Try the Re=2000 (unsteady solver) and compare to the experimental images. You could have trouble getting it to solve. A Reynolds number this high is pretty close to the edge as how high you can go without getting in too deep! It might take several minutes to solve. Don’t bother with Re=10,000.
- Save the model for the next problem.

IV. COMSOL FLUID FLOW WITH MASS TRANSFER

1. Flow over a cylinder with mass transfer.
   - Start with opening the model from the fluid flow problem. Set Re=10 and the solver to Stationary
   - Click solve, just to make sure you are back to where you started and the flow problem works as before.
   - On the left menu, click on ”Component” and then at the top of the Comsol window find ”Add physics”.
   - Add, transport of diluted species, to this component.
   - On the left menu, click on ”Transport Properties 1” which is under the tree ”Transport of diluted species”
   - On the second menu from the left, find ”Velocity Field” and change to ”Velocity Field (spf)”. This will use the computed velocity field from the fluid problem in this problem.
   - Change the Diffusion coefficient to 0.01. Since we are working in dimensionless terms this will correspond to a Peclet number of 100.
   - Right click on ”Transport of Diluted Species” in the menu tree and add a concentration boundary condition.
   - Right click on ”Transport of Diluted Species” in the menu tree and add another concentration boundary condition.
   - Right click on ”Transport of Diluted Species” in the menu tree and an outflow condition.
   - On one of the concentration boundary conditions, ”Concentration 1”, click on it in the menu tree and add the left boudary at x=0 to be c=0. Be sure to check the box that says ”Species, c”.
   - On the other concentration boundary conditions, ”Concentration 2”, click on it in the menu tree and select all the boundaries of the circle to be c=1.
   - On the outflow boundary conditions, click on it and select the upper, lower, and rightmost boundaries.
   - Cross your fingers, and click ”solve”.
   - Right click on ”Results”. Add a 2D plot group.
   - Right click on newly added ”2D plot group” and select ”Surface”
   - Click on the newly added surface plot. In the ”Settings” menu listing change ”Expression” to c. Click plot, and you should get the concentration field.
   - Change Re=10 and Peclet=1000 (Diffusion coefficient =0.001) and resolve. Change to Pe=10 (D=0.1) and Pe=1 (D=1). Observe what happens.
   - Put Pe=100 and change from Re=10 to Re=100 and resolve.
   - Make a 9x9 table and include a snaphot of the concentration field at each. Re=1 and Pe=10, 100, 1000. Re=10 and Pe=10, 100, 1000, Re=100 and Pe=10, 100, 1000