Fluid motion

I. COMSOL

1. Flow over a cylinder in free stream of uniform flow. Go back to your model from last week. Adjust the Reynolds number to 1; just to get a nice and stable solution to start with. Adjust the boundary conditions. The inlet should be uniform flow with x velocity of 1. Outlet on the right should be constant pressure. The upper and lower boundaries should have a "moving wall" with a velocity of 1. Make the region of flow that you are solving go from $-5 < y < 5$ and $-10 < x < 20$. Center the cylinder with a diameter of 1. Adjust your comsol simulation to compute the net force on the object (I can show you how to do this in class - but add "derived values" in results, select line integration, select the walls of the cylinder, click on replace expression, under component 1, laminar flow, auxiallary variables, total stress - select normal stress x-component. Convert this number to a predicted drag coefficient - being careful of the units. You can either run different Reynolds numbers by hand, or learn to set up a parametric sweep (again, I can show you). Either way, you will want to space your points in Re logarithmically. Compute drag as a function of Reynolds number using the stationary solver from Re=0.01 to Re=200. Compute drag as a function of Reynolds number using the transient solver from Re=50 to Re=2,000. You don't need to go overboard - just select 10 values of Reynolds number or something along those lines. Create a single plot of drag coefficient versus Reynolds number and compare to the experimental data. You can look up the result online to compare.

2. No consider a ellipse shape for the problem above where the long end is pointed toward the flow direction. Make the ellipse have a 0.5 b-semiaxis and 5 for the a-semiaxis. Recompute the problem above. Make a plot of drag coefficient versus Reynolds number and overlay the result of the cylinder. What conclusion can you draw about the elongated shape at low and high Reynolds number?

3. For flow in a circular pipe, Poiseuille's Law which relates applied pressure drop to volumetric flow rate is,

   $$\Delta P = \frac{128\mu L}{\pi D^4} Q$$

   This solution is valid for laminar flow when $Re < 2000$. First, compute flow in pipe in comsol and confirm that your result is in agreement with this law.

   For different cross sectional shapes, the basic form of the law holds but with different numerical correction factors. Using the hydraulic diameter, $D_h = 4A/P$ (where A is the cross-sectional area and P is the perimeter) we can write the law for different cross sectional shapes as,

   $$\Delta P = C \frac{128\mu L}{\pi D_h^4} Q$$

   where C is a numerical constant you can compute in Comsol. Find C for a square, a 2x1 rectangle, a 4x1 rectangle, and an equilateral triangle cross sectional shape.

4. Consider flow in a tube. At low Reynolds number, a neutrally buoyant particle placed at the the tube entrance will "go with the flow". Whatever streamline the particle starts on, the particle will stay on forever. Nothing interesting happens. It is a curious fact that a particle in a tube at "moderate" particle Reynolds number (i.e. greater than 1, but still laminar flow) will not stay on the streamline it starts on. As the particle traverses the tube, it will feel one force which will push it toward the wall and another that will repel it. The particle will then assume a fixed radial location in tube. If the particles come into the tube in random radial locations, the particles leave the tube confined to an annular ring; all particles are at the preferred radius.

   Let's explore whether this basic effect can be determined via comsol. Again, in the interest of computational time, let's do the problem in 2D. Set up a dimensionless channel of height 2 (from $-1 < y < 1$). Set the length to 20 units. Place the particle (a circle) in the middle of the channel with respect to x. Set the particles diameter to 0.1 to start with. Set the inlet velocity field to be $u(y) = 1 - y^2$. Set the outlet to be zero pressure. The upper and lower boundaries should be solid walls with no-slip. Set the viscosity to 0.01 such that the channel Reynolds number is 100 and the particle Reynolds number is 10. Compute the x and y component of the drag force on the circle. The result for the x force should be different than in problem 1 since the flow is confined in a channel. Compute the y component of the drag force as a function of y location of the particle. What does this analysis say the equilibrium position of the particle would be? Can you think of an effect (that would exist in our 2D problem as well as the 3D problem) that is not addressed by the current analysis?

II. HAND CALCULATION

Normally in fluid dynamics we assume the no-slip condition at a solid wall. This condition is empirically true for most cases, though there are exceptions where slip has been observed. An interesting case where there is "apparent" slip is when you have charged walls, ions in the fluid (like a dissolved salt solution), and applied electric fields.

A material like glass will typically become negatively charged when the solid comes in contact with water. This negatively charged wall will attract positive ions from the aqueous salt solution to screen the surface charge. However, molecular thermal motion will keep the ions fluctuating such that a diffuse layer of positive charge will form in the liquid. This layer will have a thickness on the order of 10 nm (that's nanometers) for a 1 mM (milli-molar) salt solution, the length scale is called the Debye length. More than 10 nm away from the wall, the solution will be neutral with the number of negative ions balanced by the number of positive.

In this case, if you apply an axial electric field down the channel, the electric field will exert a force on the positively charged ions near the wall and the resulting force will drag the fluid along. This electrical force will only be exerted in this charged layer which is 10 nm thick. Thus, even though the fluid at the wall will obey the no slip condition, the fluid 10nm away will be in motion. The net result after is that the flow in a channel which is much larger than 10 nm will have an apparent slip velocity at the wall. This slip velocity is proportional to the electric field. The formula for the slip is $U_{slip} = -\epsilon_0 \epsilon_w \zeta E/\mu$, where ϵ_0 is the vacuum permittivity, ϵ_w is the relative permittivity for water (approximately 80), E is the applied electric field, μ is the dynamic viscosity, and ζ is the so-called zeta-potential, the voltage difference between the wall and the solution far away. In typical glass/water systems $\zeta \approx -100$ mV. This theoretical result is over 100 years old, it is called the Helmholtz-Smoluchowski equation. The effect, called electroosmosis has been known experimentally for over 200 years and was first observed in porous clays. Sometimes it is more convenient to measure the combination of parameters and call it the electroosmotic mobility, $b = -\epsilon_0 \epsilon_w \zeta/\mu$, and simply express the slip velocity as $U_{slip} = bE$.

If we have a long, thin tube with no applied pressure difference and the electroosmotic (EO) slip velocity is a constant, you would find trivially, that the solution to the Navier-Stokes equation is a constant fluid velocity. All the fluid moves down the tube at the slip velocity. In recent years, these electroosmotic flows have been exploited in a number of modern micro and nanoscale systems. Given the results for pressure driven flow, the hydraulic resistance scales poorly as the channel or pipe becomes very small. Thus in small systems, electroosmosis is a practical and efficient pump. It is also a simple pump because it has no moving parts, just apply a voltage across your channel.

While all this background may sound complicated, the final result is quite simple. From the known (or measured) system parameters we can compute the slip velocity. We then solve the flow problem using the slip velocity as the boundary condition for the Navier Stokes. At a solid surface, the normal component of the velocity is zero while the tangential velocity is given as the slip value.

OK. That was all background. Now the actual problems.

- Consider a small tube of radius r and length L. Assume a pressure ΔP and voltage ΔV difference are applied across the tube. The tube is long such that we can consider the flow invariant in the axial direction. The pressure gradient $dP/dx = -\Delta P/L$ and the electric field $E = -\Delta V/L$ are constants along the length of the channel. We will assume the electroosmotic mobility is a known constant (usually comes from measurement). Thus, the apparent slip velocity at the wall has a known value U_{slip}. Solve the Navier Stokes equations for this case. State the velocity profile as a function of r and how it depends on the system parameters, $r, L, \Delta P, \Delta V, \mu$, and U_{slip}. Compute the average fluid velocity i.e. $U_{ave} \pi r^2 = Q$ where Q is the total volumetric flow rate. Write your expression for the average velocity in the form, $U_{ave} = A\Delta P + B\Delta V$, where A and B are constants that you will figure out. If you are really smart you can use superposition and look at the solution for pressure driven flow in a pipe (section 8.9 of the book).

- Consider a tube of radius 10 microns and length 10 mm, containing an aqueous solution. The measured mobility is $b = 7 \times 10^{-8} \frac{\mathrm{m}^2}{\mathrm{V \cdot s}}$ and the electric field is in Volts/m. The applied voltage is 5 kV. Compute the mean flow velocity if there is no applied pressure gradient. Compute the applied pressure needed to obtain the same mean flow velocity if there were no applied electric field. Repeat the above calculations if the tube has a radius of 1 micron.

- In some applications, we can intentionally coat or modify the solid surface to change the surface charge and thus change the apparent slip velocity. Consider a case where the channel from $0 < x < L_1$ has slip velocity $U_{s,1}$ and $L_1 < x < L$ has slip velocity $U_{s,2}$. Consider flow in such a system where we have two regions of two different slip velocities, but NO applied pressure. Since the channel is long compared to the other scales, we can assume everything is uniform in the axial direction and simply assume we have two channels which are connected together in series.

Since the slip velocity is different in the two regions, but the total flow through any cross section of the channel must be the same, something has to give. What happens is that while the net applied pressure is zero, each region will have to acquire an internally generated pressure gradient. This pressure gradient will enhance or retard the electrically driven slip flow such that each region has the same total flow rate. The pressure gradient in each region is a constant and described as $dP/dx|_1 = -\Delta P_1/L_1$, and $dP/dx|_2 = -\Delta P_2/L_2$ where the total applied pressure is zero, $\Delta P_1 + \Delta P_2 = 0$. Due to continuity of electrical current, the electric field is a constant in both regions, $E = E_1 = E_2 = \Delta V/L$. The average flow in both regions must be equal, $U_{ave,1} = U_{ave,2}$. Use all the above relationships as well as your formula from part 1, $U_{ave} = A\Delta P + B\Delta V$, to determine the pressure gradient in each region.

- Assume the following parameters. A tube of radius 10 microns and total length 10 mm. The electroosmotic mobility is measured to be $b = 7 \times 10^{-8} \frac{\mathrm{m}^2}{\mathrm{V \cdot s}}$ in region 1. In region 2 it is reduced to $b = 0.7 \times 10^{-8} \frac{\mathrm{m}^2}{\mathrm{V \cdot s}}$. The applied voltage is 5000 Volts. Consider cases for $L_1/L_2 = 1/4$, 1, and 4. Compute the average fluid velocity in both regions for all cases of L_1/L_2. Note that the average velocity in region 1 and region 2 better be equal in each case. Check your result and make sure that the limiting cases work, i.e. L_1 and L_2 going to zero limit to the result you derived in part 1.