Fluid motion

I. REYNOLDS NUMBER ESTIMATES

Estimate (order of magnitude) the Reynolds number for

- A car traveling at 55 mph.
- A passenger plane at cruising speed.
- A hurricane.
- A bacteria in water (2 μm in size moving at 100 μm/s)
- Make up 2 other things that involve fluid flow and estimate the approximate Reynolds number.

II. COMSOL

In this homework we will use Comsol to model some flow problems. The point is to get you a little more familiar with finite element modeling and gain a little intuition about flow problems.

Feel free to explore things that seem interesting to you. Keep what you turn in simple. Create a plot or two and provide a little explanation of what the results mean. Keep your geometry scaled so that the length scale is 1 unit, the density is 1 unit, and the viscosity is the inverse Reynolds number. For reasons we will discuss in class you can expect the software to break down once the Reynolds number exceeds a few hundred or maybe 1000, depending on the problem. In all cases keep the problem as steady flow (stationary).

You can work on these problems with a partner and turn in one joint assignment if you would like.

1. Create a 2D channel which is 1 units high (in y) and has a length of 20 units (in x). Set the inlet corner for the rectangle at (0,0). Specify the inlet x velocity field at $x = 0$ to be $u = 4y(1 - y)$. The parabola is an exact solution to the Navier Stokes equations. Set the outlet condition at $x = 20$ to be at a pressure of zero. Set the viscosity (Reynolds number) to 1 initially. When you run this simulation, it should be pretty simple and show the parabolic velocity profile the whole length of channel. Use the simulation to measure the pressure drop across the length of this channel. Create a plot of pressure along the channel's centerline as a function of x. Change the Reynolds number (viscosity) and see how the pressure drop changes (holding everything else constant).

2. Add an obstruction to your channel in the above simulation. Make a block that is 1/2 a unit high and one unit long and add it to your channel at x=5. Run the simulation at a few different Reynolds numbers and look at the flow field and the streamlines. Observe what happens qualitatively to the flow as Reynolds number increases. Pay attention to when you observe clear breaking of the fore-aft symmetry. Measure the pressure drop as a function of the Reynolds number and compare to the case with and without the obstruction.

3. Consider flow over a cylinder. Place a cylinder with a diameter of 1 inside a 2D channel. Center the cylinder at the origin. Make the channel height go from plus and minus 3. Make the length sufficient such that entance/exit effects are not important (20 units should be OK). Set the inlet velocity on the left to be $u = (9 - y^2)/9$ - our parabolic velocity profile with a velocity of 1 along the centerline. Plot the streamlines at different Reynolds numbers. Look up the result for flow over a cylinder in a uniform free-stream flow and see if your results make sense in general.

   Compute the total force acting on the cylinder. Compute the total force, by selecting "derived values", "line integration". Select the surface of the cylinder. Under "expression" look for "total stress, x-component". Adjust the Reynolds number and recompute the drag. Start low, (like 0.01) and then increase up until a few hundred (where the solution will start to break down). Make a plot of drag coefficient versus Reynolds number. Compare your solution to that of a cylinder in a free stream which is easily found online.

4. In the previous problem offset the cylinder from the centerline. Compute the lift force which would be pushing the cylinder toward or away from the wall. Generate a plot which shows the lift force as a function of y position of the cylinder in the channel. See how this lift force profile changes for different Reynolds numbers.

5. Flow in curved pipes can be very different than in straight ones. The curvature can cause a secondary flow called Dean flow. The term secondary means that the fluid flow is still mostly down the length of the pipe, but if you subtract off this primary flow then you see the weaker secondary flow.

   One way to see this secondary flow is in a 3D simulation. Under the geometry creation in 3D Comsol you have the option to create a helix. Create a 1 turn helix with very little axial pitch (it will not let you define the pitch as zero). You can control the radius of the tube and the radius of the helix. Start with a low Reynolds number (viscosity). Define one end of the helix to be an inlet with a uniform velocity of 1 and the other end to the outlet with zero pressure.

   To visualize the secondary flow, try taking a planar slice aligned with the coordinate axis. If you take the right slice, the velocity components across a cross section of the pipe will be perpendicular to the main flow. You can then look at the velocity field and streamlines in this planar slice to see the secondary flow.

   Explore the behavior of the secondary flow qualitatively. Try adjusting the Reynolds number and see what happens (just don't go too high). Look up what Dean flow looks like online and see if your solution seems reasonable.