PH419: Physics of Biological Systems
Assignment 4
Fluid Flows in biology

1. **E. coli swimming**

   E.coli swims at about $20\mu m/s$ by rotating a bundle of helical flagella. If the motors were to turn 10 times faster than normal, what would their swimming speed be? If their fluid environment were made 10 times more viscous, but the motors were to turn at the same rate, what would the swimming speed be? How does the power output of the motor change in these two hypothetical situations?

2. **Blood flow**

   The left ventricle of the human heart expels about $50cm^3$ of blood per heartbeat. Assuming a pulse rate of 1 heartbeat per second and a diameter of the aorta of about $2cm$, what is the mean velocity of blood in the aorta? What is the Reynolds number?

3. **Protein centrifugation**

   Proteins and other macromolecules can be separated by size using centrifugation. The idea is to spin a sample containing proteins of different size in solution. The spinning produces a centrifugal force per unit mass g_c , which leads to diffusion with a drift velocity that depends on the protein size. We assume that a protein in the sample can be approximated as a ball of radius R. See the lecture notes for details.

   (a) What is the drift of a protein as a function of its radius? Assume that ρ_p and ρ_s are the protein and solvent densities, and η is the viscosity of the solvent.

   (b) Estimate the drift velocity for hemoglobin in water in an ultracentrifuge with $g_c 10^5 g$, where $g \approx 10m/s^2$ is the acceleration of freely falling objects in Earths gravitational field. Assume a typical protein density of $1.2g/cm^3$.

   (c) We would like to separate two similar proteins, having the same density, $\rho^{'} = 1.35g/cm^3$. They have diameters of $4nm$ and $5nm$, respectively. The two protein species start out mixed together in a thin layer at the top of a 1 cm long centrifuge tube. How large should the centrifuge acceleration g_c be so that the two proteins are separated before they drift to the end of the tube?

4. **Hydrostatic pressure**

   Consider a fluid at rest. The forces on the fluid are due to the fluid pressure p and the gravitational pull of the earth.

   (a) What does the Navier-Stokes equation reduce to in this case?

(b) Solve the Navier-Stokes equation to determine the pressure in the fluid by assuming a uniform fluid density. The gravitational force can be taken to be along the negative $\hat{\mathbf{z}}$ direction.

(c) Most of the mass of the Earths atmosphere is contained within the troposphere, which is approximately $10km$ high. The density of air is roughly $1kg/m^3$, and the acceleration of gravity is roughly $10m/s^2$. Estimate the atmospheric pressure.

(d) If you raise your arm above your head as the doctor is measuring your blood pressure, how much will the measurement change compared with when you keep your arm level with the heart. Assume the height difference is $15cm$, and further that the density of blood and water are the same.

5. **Simulations of the Navier Stokes Equation**

   (NB.*** This is optional. Also, if you try it, it's not time limited. You can submit the plots and code whenever you finish them.)

   For most realistic scenarios and boundary conditions, it is not possible to obtain analytic closed-form solutions of the Navier-Stokes equations. In such situations, one resorts to numerical solutions of the fluid dynamics equations.

   OpenFOAM (https://openfoam.org/) is a free, open-source software for computaional fluid dynamics. You can download and install the software package from:
   https://openfoam.org/version/6/

   The user guide for using the software package is available at:
   https://cfd.direct/openfoam/user-guide/

   (a) As a first exercise, consider the problem of a shear flow for a fluid in a cavity with a constant velocity of the top wall. This is a tutorial problem with the codes for simulating in OpenFOAM available at:
   https://cfd.direct/openfoam/user-guide/v6-cavity/#x5-40002.1
   Following the tutorial, generate the pressure gradient and velocity vector plots for the fluid in the cavity.

   (b) Now consider the case of flow of an incompressible fluid around a cylinder. Plot the fluid flows for a range of Reynolds numbers, starting from $Re \ll 1$ to $Re \gg 1$. Also, verify whether changing the velocity and the viscosity such that the Reynolds number stays constant changes the nature of the flow.
   A paper describing the simulation is uploaded along with this assignment.

   (c) Simulate the flow around a sphere contained in a cylindrical pipe. Generate the velocity fields for the case of (i) a linear flow field, and (ii) a rotational flow field. How does the flow change with Reynolds number, and with changing diameter of the pipe?