Particle dynamics class, SMS 618, Emmanuel Boss

Fluid flow and transport around a translating sphere, using a Matlab environment produced by U. H. Thygesen and T. Kiorbe.
Source code+documentation: http://www.dfu.min.dk/uht/Software/Default.htm

Today you will run a program that computes the flow fields around a translating sphere at low Reynolds number and the solute field around it. The program is applicable to other problems which we will ignore today but may be of interest to you.

A numerical model such at the one we use today uses a finite grid around the cell to solve the differential equations for the flow and solute concentration. In general, the finer is the grid the more accurate are the results (a derivative is calculated exactly in the limit of small spacing). However, the finer the grid the longer the time it takes to execute the computation. One should always test for sensitivity of model result to the grid details (e.g. by doubling the grid cells).

You begin the program by typing snow on the Matlab command line.

a. On figure 1, specify the grid parameters. In all calculation the azimuth angles is neglected due to symmetry.

- No. of angles-number of angular grid points.
- No. of radii-number of radial grid points.
- Max. radius-Limit of computational where boundary conditions are applied.
- 0<Gamma<1-assymetry parameter allowing for denser grid points in the particles wake (where the action is).

The model is non-dimensionalized with length-scale, x, y=(x', y')/r_0 (e.g. x=1 means one radius away from the origin in the positive x-direction), and U the velocity scale. Concentration is non-dimensionalized with c=C(r_0)-C(∞).

b. On figure 2, specify the parameters for fluid flow computation and visualization.

- For today's lab we will specify a 'Numerical N-S solver' at the top box.
- Choose the Reynolds number 'Re='. Keep Re<20 (the higher the Re the longer it takes and the finer the needed grid).
- The rest of the parameters are plotting parameters (which can be varied after solution).
- Hit 'solve' to derive the flow field.

c. On figure 3, specify the Peclet number $Pe=Re*D/\nu$, where D is the diffusivity of the solute and ν the kinematic viscosity of water. For micro-nutrients such as Nitrate, $D/\nu \sim 1000$.

- Specify 'Lambda'=1. $\lambda<1$ provides a parameterization of diffusion in the presence of turbulence.
- Inner BC-Specify Dirichlet (fixed concentration at the sphere). Neumann specifies a constant flux from the boundary.
- 'Normalize'-Specify BC.
- Method- Specify 3-4 order upwind.
- The rest of the parameters are plotting parameters (which can be varied after solution).

The last window, in figure 4, contains several parameters regarding the solution:

- First, specify a threshold concentration ($C_{threshold} <1$) for plume calculations.
- Given this threshold it provides the width (non-dim with r_0), cross-sectional area (non-dim with r_0^2), and volume of plume (non-dim with r_0^3). Net solute flow is also computed (non-dim with cr_0^2U).

Class assignment:

The pure diffusive flux (in the absence of flow) is given by: $F=4\pi D\{C(r_0)-C(\infty)\}r_0$. The flux in the presence of motion, $F_{Pe}=\{C(r_0)-C(\infty)\}r_0^2UF'=PeD\{C(r_0)-C(\infty)\}r_0F'$, where F' is the output in figure 4. Thus, the Sherwood number is given by $Sh= F_{Pe}/F= F'Pe/4\pi$.

1. For Re=0.001, 0.01, 0.1, 1, 10 (if you make it, a choice of [40,80,20,0.5] in figure one, worked for me) and Pe=Re*1000, compute the Sherwood number (Sh). How does is it vary with Re? What does it imply about the change in rate of solute transport as function of size ($Re=UL/\nu$, and $U \propto L^2$) for the same type of particles?

2. For Re<<1 (Stokes Solution), compute the Sh number for Pe=0.001, 0.01, 0.1, 1, 10, 100, and 1000 (note, you may have to have max radius as much as 50 or more to resolve the plume with High Pe). How does it compare with the results from the literature provided in the class notes? How does the volume of the plume vary with Pe?