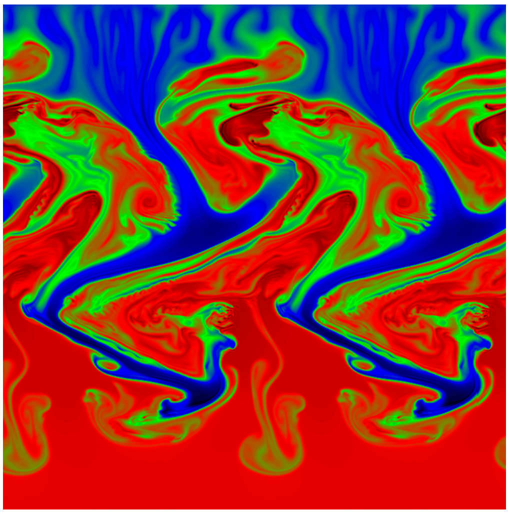
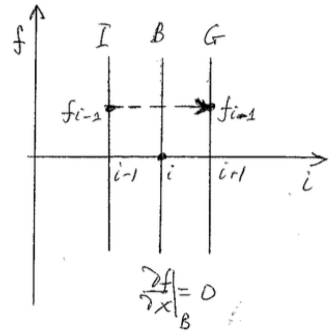
By 28th of April:

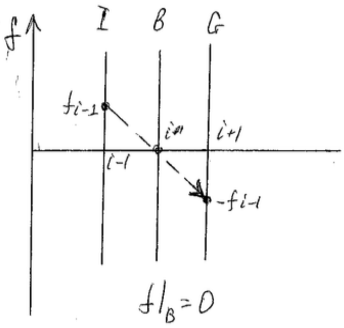


(1) Fun with ~~flags~~ boundary conditions, gravitational stratification, and Rayleigh-Taylor instability. Modify your 2D Lax-Friedrichs hydrodynamic solver to accommodate both periodic and closed/reflective boundary types. The closed BCs at the boundary are defined as follows:

$$\frac{\partial \rho}{\partial x}=0;\ \frac{\partial v_{\parallel}}{\partial x}=0;\ v_{\perp}=0.$$

Remember that your grid is node-centered (the fluid parameters are measured at the grid nodes), so, visually, your boundary condition looks like:





Add the gravitational sources to the right-hand sides of the Navier-Stokes equations so they read as:

$$\frac{\partial \rho}{\partial t}+\nabla\cdot(\rho\mathbf{v})=D(\rho),$$
$$\frac{\partial(\rho\mathbf{v})}{\partial t}+\nabla\cdot(\rho\mathbf{v}\mathbf{v})+\nabla p=\mathbf{D}\left(\rho\mathbf{v}\right)+\rho\mathbf{g},$$
$$\frac{\partial \varepsilon}{\partial t}+\nabla\cdot(\mathbf{v}(p+\varepsilon))=D(\varepsilon)+\rho\mathbf{g}\cdot\mathbf{v}.$$

Use γ=4/3. If you are brave enough, build 4th order Runge-Kutta solver and enjoy larger time step and lower diffusivity, leading to sharper features. Use constant gravity acceleration **g**, direct it along one of the axes, which will make the problem simpler without loosing generality. Note, when you add gravitation in the problem, you remove the symmetry and define your "vertical" and "horizontal" directions, as well as "top" and "bottom". Set the horizontal physical domain size -0.25…0.25 and vertical domain size -0.75…0.75, and set the resolution so $\Delta x=\Delta y$. Make the side boundaries periodic, and top and bottom boundary (as defined by the sign and direction of your gravity acceleration vector) closed. As the initial condition, use

$$h<0:\rho=1;$$
$$h\geq 0:\rho=2;$$
$$p=2.5+\rho g h,$$

where h is height. Note how your initial condition looks like: a heavier fluid is on top of a lighter fluid, but still in the pressure equilibrium, so it is the exact solution for the hydrostatic equation $\nabla p=\rho\mathbf{g}$. To perturb the initial condition use a random velocity perturbation with the amplitude 0.01.
Plot the evolution of the solution (density, velocity, internal energy, pressure) on time. Make a movie, if you can. Some time well into the simulation, you will have a lot of mushrooms like in the picture above, where the density is shown, taken at ~70000'th timestep.
Study how the instability develops by looking at the evolution of average quantities, such as maximum velocity, total energy, total kinetic energy, on time.