Physics 335 | March 30, 2023
Physics 635 | **Due**: April 16, 20 & 25, 2023

Homework #10

You may work together for this homework.

Project #2

Solve the coupled quasi-linear elliptic equations for stationary flow of an incompressible fluid.

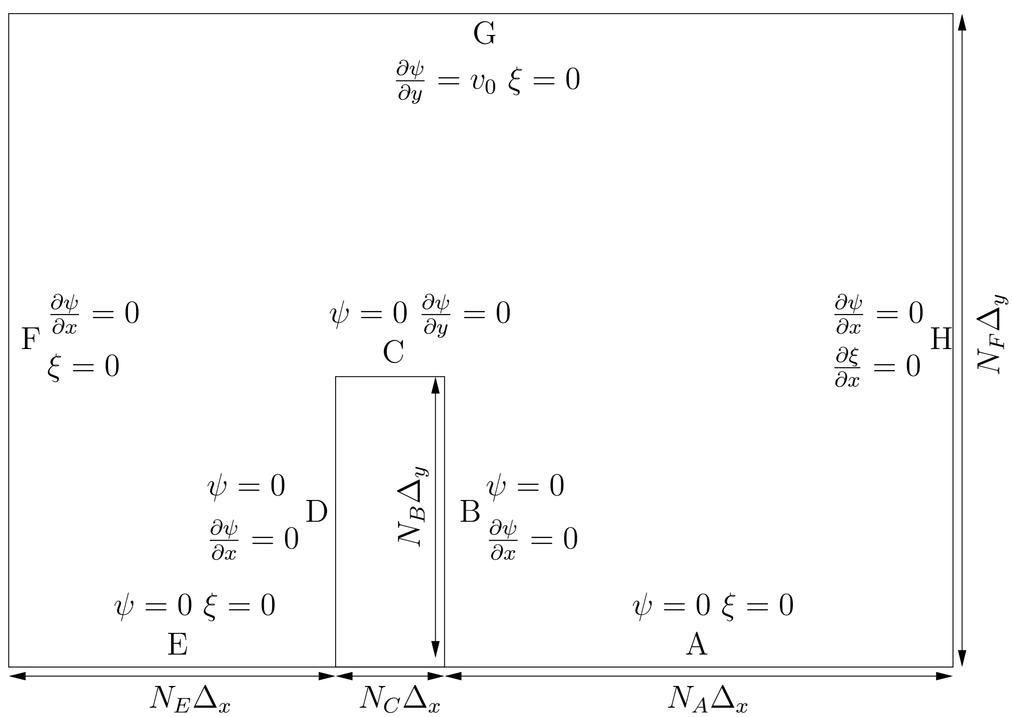
As we have seen in class, the flow of an incompressible, viscous fluid in two dimensions, expressed in Cartesian coordinates, is governed by a set of two coupled elliptic equations for the *stream function* ψ,

$$\nabla^2\psi = -\xi \tag{1}$$

and the *vorticity* ξ,

$$\nabla^2\xi = \frac{1}{\nu}\left(\frac{\partial\psi}{\partial y}\frac{\partial\xi}{\partial x} - \frac{\partial\psi}{\partial x}\frac{\partial\xi}{\partial y}\right). \tag{2}$$

Solve these equations using SOR for a fluid flow past a plate using the boundary conditions diagrammed below as we discussed in class.



You can experiment with various different geometries and parameters. You should use vertex centered differencing and locate the various boundaries at cell faces. In the x-direction, you need to make sure that the lengths of faces `A`, `C`, and `E` are integer multiples of some discretization length Δ_x. Similarly, the lengths of `B` and `F` must be integer multiples of some discretization length Δ_y.

As an example, you could set the length of `C` to $L_C = 1$ as a fundamental scale for the problem, and then set $L_B = 3$. Then, letting $L_A = 6$, $L_E = 5$, and $L_F = 9$ fully specifies the computational domain in "arbitrary dimensions". To fully specify the problem, we could also specify $v_0 = 1$ and $\nu = 0.5$. A low resolution solution might have 2 zones across `C` yielding a 25×19 grid. A good solution might have four times as many points in each direction.

In my implementation, I start with $\psi = v_0 y$, $\xi = 0$. I then update ψ over the entire grid using overrelaxation with a parameter ω. I then update ξ, again over the entire grid using overrelaxation with the same parameter ω. This completes one "sweep". In the next sweep you start updating ψ again, this time with the improved value of ξ.

After each sweep, compute the residual

$$R_\psi(x,y) = \nabla^2\psi + \xi \qquad R_\xi(x,y) = \nabla^2\xi - \tfrac{1}{\nu}\left(\tfrac{\partial\psi}{\partial y}\tfrac{\partial\xi}{\partial x} - \tfrac{\partial\psi}{\partial x}\tfrac{\partial\xi}{\partial y}\right) \tag{3}$$

and their L_2-norms

$$\|R_\psi\| = \left(\int R_\psi^2 dxdy\right)^{1/2} \qquad \|R_\xi\| = \left(\int R_\xi^2 dxdy\right)^{1/2} \quad . \tag{4}$$

It can be very useful to look at contour plots of R_ψ and R_ξ. If they much larger in some places than in others (for example on boundaries), then this indicates that there is a bug, and you can even take a guess where.

Make a plot of $\|R_\psi\|$ as a function of number of sweeps for different values of the relaxation parameter ω. The optimal choice for ω will depend on several different parameters, including the viscosity ν.

You can visualize your solution in various different ways. Contour lines of ψ, by construction, mark stream lines of the fluid (see the MatLab routine `contour`). You can also compute the fluid velocities v_x and v_y from ψ and plot those (see the MatLab routines `gradient` and `quiver`).

This is a pretty open-ended project. You can vary the resolution and the location of the outer boundaries to make sure that your solution is not affected by those choices (if you place the downstream boundary too close to the plate, for example, you will see large but spurious eddies developing). To study physical effects, you can increase the velocity V_0 or decrease the viscosity ν, both of which have the effect of increasing the so-called *Reynolds number*. For large enough Reynolds numbers, you may run into instabilities, especially at the downstream boundary. *Hint: Code the ξ boundary condition on surface* `H` *directly as a* 1^{st}*-order condition instead of incorporating it into the PDE. This will postpone the onset of this instability. It is OK, for this project, to treat all of the boundaries this way.*