ME/AE/BME 518 - Computational Fluid Dynamics

Spring 2019, Project 2, Due February 12

1 Background

Now that you have survived the first project, you already have a code that solves the Laplace's equation in a Cartesian domain. In this assignment, we will add another governing equation to the set and will turn it into a Navier-Stokes solver and will investigate laminar flow in a lid driven cavity. The two-dimensional incompressible Navier-Stokes equations can be written as

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1}$$

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{2}$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \nu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \tag{3}$$

where u and v are the velocities in the x and y-directions, respectively; ρ is the fluid density; ν is the kinematic viscosity; and p is the pressure. These three equations can be reduced down to two equations if one uses the vorticity ω and the streamfunction Ψ as the dependent variables.

In the first part of this project, **show that the governing equations can be re-written as**

$$\frac{\partial \omega}{\partial t} + u\frac{\partial \omega}{\partial x} + v\frac{\partial \omega}{\partial y} = \frac{1}{Re}\left[\frac{\partial^2 \omega}{\partial x^2} + \frac{\partial^2 \omega}{\partial y^2}\right] \tag{4}$$

$$\frac{\partial^2 \Psi}{\partial x^2} + \frac{\partial^2 \Psi}{\partial y^2} = -\omega \tag{5}$$

where ω is the vorticity, Ψ is the streamfunction and u and v are the Cartesian velocities defined as

$$u = \frac{\partial \Psi}{\partial y}, v = -\frac{\partial \Psi}{\partial x}$$

Note that you need to nondimensionalize your equations by defining appropriate reference quantities.

Next, consider the square lid-driven cavity problem [see Fig. (1)] represented by the equations given above. Note that the non-dimensional length of each side of the cavity is 1.0 and the lid velocity of the fluid is prescribed. As can be seen, the governing equations [Eqs. (4) and (5)] are coupled with the vorticity equation involving an unsteady term $\partial\omega/\partial t$. Traditionally, the governing equations are integrated using time-accurate time-domain techniques. In this project, you will use the Implicit Euler method. The spatial discretization should be performed using second-order central differences. To keep things simple, lag the velocity terms in the vorticity transport equation for one time-step. That means, use a semi-implicit approach for this project. We will talk about the advantages of this approach in class.

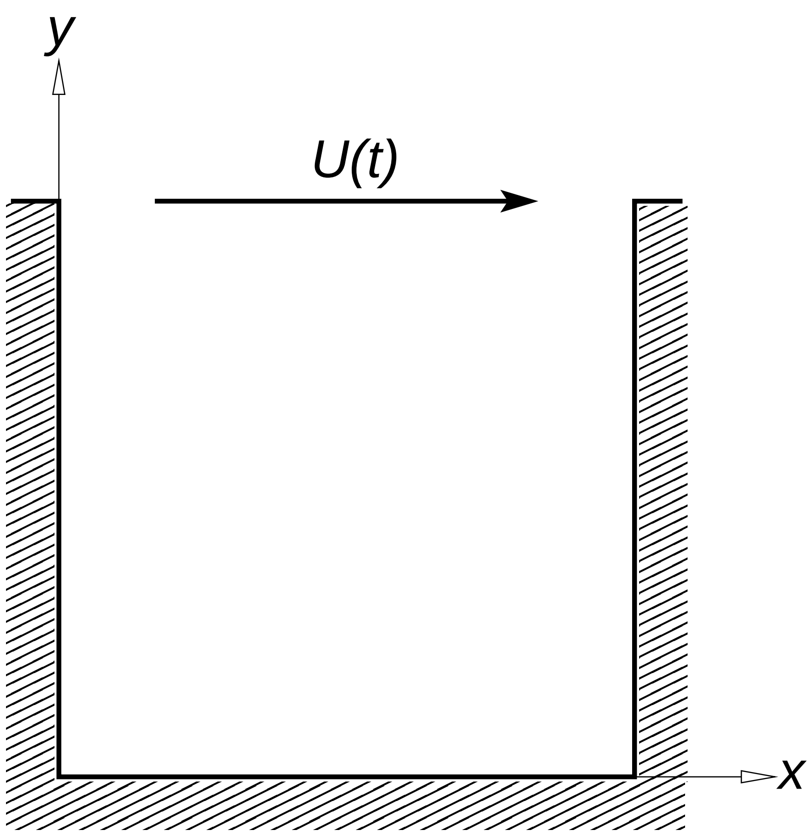

Figure 1: Lid-driven cavity geometry.

The following boundary conditions can be used in this project. First, the values of streamfunctions are set to a constant value ($\Psi = 0$) value on the cavity boundaries. This ensures the continuity of the streamfunction values on the corners of the lid. It also satisfies the no-slip boundary conditions on the solid walls (Think about how?). Second, the vorticity on the boundaries is determined from streamfunction values at the inner nodes using a second-order finite-difference approximation.

As your second task, show that

$$\omega(x,1) = -\frac{[4.0\Psi(x,1-\Delta y) - 0.5\Psi(x,1-2\Delta y)]}{(\Delta y)^2} - 3.0\frac{U_{lid}}{\Delta y} \tag{6}$$

Also determine the vorticity boundary conditions on the solid walls.

Your third task is to model the flowfield and obtain steady-state numerical approximations for various Reynolds Numbers (10,100,500) and a lid velocity of 1.0 ($U_{lid} = 1.0$. In your computations, use grid resolutions of 51×51, 101×101, and 201×201. For these grids, determine the truncation error and the order of your method by carefully observing solutions for successive grids. Plot your convergence history (Log10(Global Residual) vs. Iteration).

Please include plots of your solution (contour plots of streamfunction and vorticity). Compare your results to those of Ghia et al. Include, tables and detailed discussion or the results in your report.

References [1] Ghia, U., Ghia, K.N., and Shin, C.T., “High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method,” *Journal of Computational Physics*,Vol. 48, No. 3, pp. 387-411, 1982.