Simulation of Lid-Driven Cavity Flow

The lid-driven cavity problem is of high importance in fluid dynamics serving as a benchmark for the validation of numerical methods as well as for studying fundamental aspects of incompressible flows in confined volumes driven by the tangential motion of one or more bounding walls. The goal of this tutorial is to simulate the benchmark case of two-dimensional flow driven by a lid in a square cavity for different Reynolds numbers using a self-developed finite difference method-based PDE solver and validate the results against the results reported by Ghia *et al.* [1].

Problem description

The problem consists of a fluid with density ρ and viscosity μ inside a square cavity. The cavity is made up of three rigid walls with no-slip conditions and length L as well as a no-slip lid which moves with a tangential velocity of U, see Fig. 1. The continuity equation and incompressible Navier-Stokes equations, given below, govern this flow:

$$\nabla \cdot \mathbf{u} = 0, \tag{1}$$

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{\nabla p}{\rho} + \nu \nabla^2 \mathbf{u}, \tag{2}$$

where $\mathbf{u}$ is the flow velocity, p is the fluid pressure, t is time. Now, Eq. 2 can be non-dimensionalized as

$$\frac{\partial \mathbf{u}^*}{\partial t^*} + \mathbf{u}^* \cdot \nabla^* \mathbf{u}^* = -\nabla p^* + \frac{1}{Re} \nabla^{*2} \mathbf{u}^*, \tag{3}$$

where Re is Reynolds number, defined as $Re = \rho UL/\mu$, which governs the nature of the flow physics.

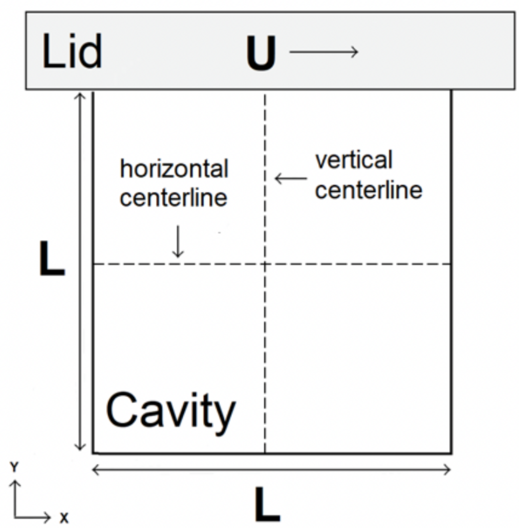

Figure 1: Schematic representation of the problem.

This represents three scalar equations, one for each velocity component (u, v, w). But we will solve it in two dimensions, so there will be two scalar equations. Additionally, there will be one equation to solve pressure. Here is the system of differential equations: two equations for the velocity components u, v and one equation for pressure:

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

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

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

Let's consider $L = 2$ units and choose appropriate values for other parameters corresponding to a specific value of Re. The initial condition is $u, v, p = 0$ everywhere, and the boundary conditions are:

$u = 1$ at $y = 2$ (the "lid");

$u, v = 0$ on the other boundaries;

$\frac{\partial p}{\partial y} = 0$ at $y = 0$;

$p = 0$ at $y = 2$

$\frac{\partial p}{\partial x} = 0$ at $x = 0, 2$

- Discretize these three PDEs using finite difference method.
- Compute the cavity flows for Reynolds numbers 100 and 1000. Compare the results with Fig. 2 [1].
- Compare the simulation results for different grid and time-step resolutions.

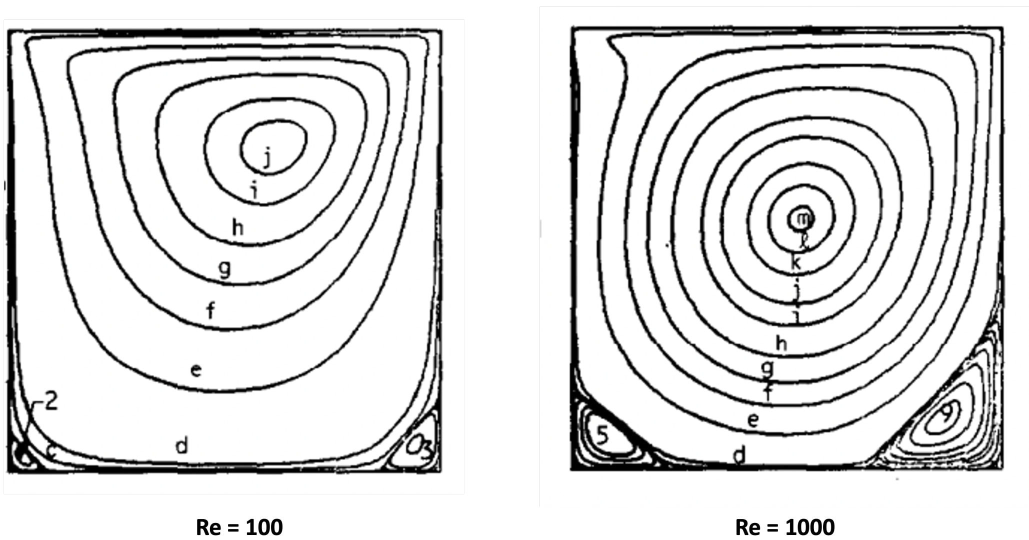

Figure 2: Cavity flow patterns for different Re values.

The code is available in the GitHub repository: Step 11: Cavity Flow with Navier–Stokes [2]
https://github.com/barbagroup/CFDPython.

Note: Interested students can also compare the results obtained using their code with the simulation carried out in open-source library OpenFOAM (Finite Volume Method). It will be demonstrated during the tutorial session if time permits. https://www.openfoam.com/documentation/tutorial-guide/2-incompressible-flow/2.1-lid-driven-cavity-flow.

References:

[1] Ghia, U. K. N. G., Ghia, K. N., Shin, C. T. (1982). High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method. Journal of computational physics, 48(3), 387-411.

[2] Barba, L. A., Forsyth, G. F. (2018). CFD Python: the 12 steps to Navier-Stokes equations. Journal of Open Source Education, 2(16), 21.