Abstract

This python code solves a problem statement known as Lid Driven Cavity (LDC) defined in the fluid dynamics field. Fluid dynamic problems can be solved using computational methods known as CFD, and the algorithm for this problem is from [Sharma,2018]. Each algorithm step is defined in code blocks with the calling of functions wherever necessary. Since the computations are mathematically intensive on arrays, the numpy library is utilized. The computational process involves multiple iterative calculations; hence the focus is on achieving computational efficiency. The computed results can be verified with benchmarked results from Ghia et. al.

Problem Statement (Example 9.1 from the book by Sharma): Lid-driven cavity flow is probably the most commonly used problem for testing an in-house Navier-Stokes (NS) solver. This is shown in the figure as a square cavity with the left, right, and bottom walls as stationary. The top wall, called here the lid, acts as a long conveyor belt and moves horizontally with a constant velocity of U0. The motion results in a lid-driven recirculating flow inside the cavity. The cavity is represented by a closed 2D Cartesian square domain of size L1=L2, with all the boundaries as the solid walls. The figure also shows the initial and boundary conditions for the non-dimensional computational set-up of the problem.
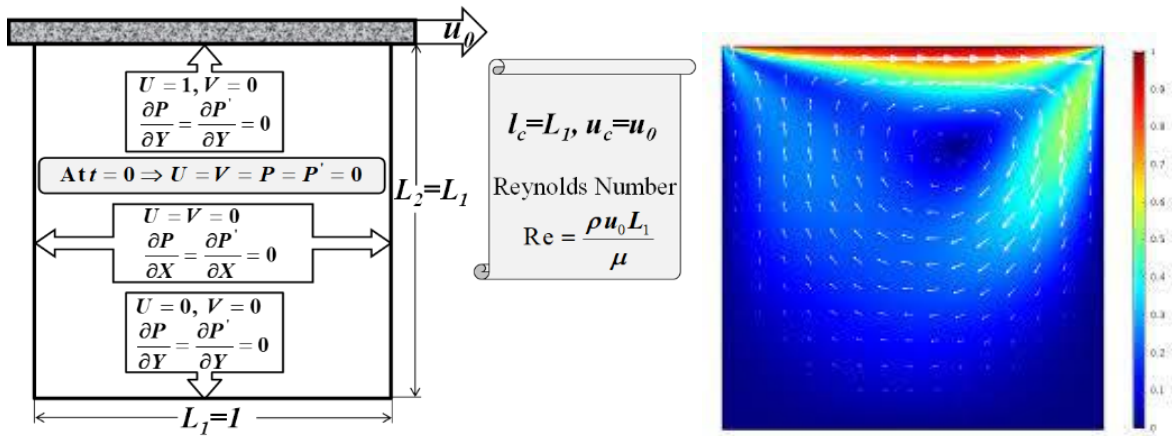

Figure 1: Computational domain and boundary conditions for the lid-driven cavity flow (left) &
Expected output flow contour from benchmark results of Ghia et al. (right)

Approach: Using the flux-based solution methodology of the CFD for the FOU scheme, develop a python program for a 2D unsteady NS solver on a *uniform* staggered grid. The length of the cavity L_1 is the characteristic length, and the lid velocity U_0 is the characteristic velocity scale. The governing parameter for the isothermal flow is Reynolds number $Re=\rho U_0 L_1/\mu$. The Re is implemented in the code by a computational set-up as $\rho=U_0=L_2=1$ and $\mu=1/Re$. After the CFD development, run the code with a convergence tolerance of $\varepsilon_{st}=10^{-3}$ for the steady-state and $\varepsilon=10^{-8}$ for the mass-conservation.

Objective: For a Reynolds number of 100 and a grid size of *42×42*, present and discuss a figure for the velocity-vector, U-velocity contour, and pressure-contour in the flow domain (3 figures).

References Sharma A., (2017), Introduction to Computational Fluid Dynamics: Development, Application, and Analysis, Ane Books Pvt. Ltd., New Delhi, Chapter 9, pp. 302-306.
Ghia U., Ghia K. N., and Shin C. T. (1982). High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method, J. Comp. Phys., vol. 48, pp. 387-411.