Cavity Flow: Numerical Simulation

Masud

October 3, 2025

Outline

Cavity Flow Problem

The lid-driven cavity flow is a classical CFD benchmark for incompressible Navier-Stokes equations.

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p + \nu \nabla^2 \mathbf{u},$$
$$\nabla \cdot \mathbf{u} = 0$$

- $\mathbf{u} = (u, v)$ is the velocity vector
- p is pressure
- ν is kinematic viscosity
- Top lid moves with constant velocity; other walls are stationary

Discretization

- Uniform Cartesian grid: x_i, y_j
- Time-stepping: explicit or semi-implicit schemes
- Finite difference / finite volume approximations:

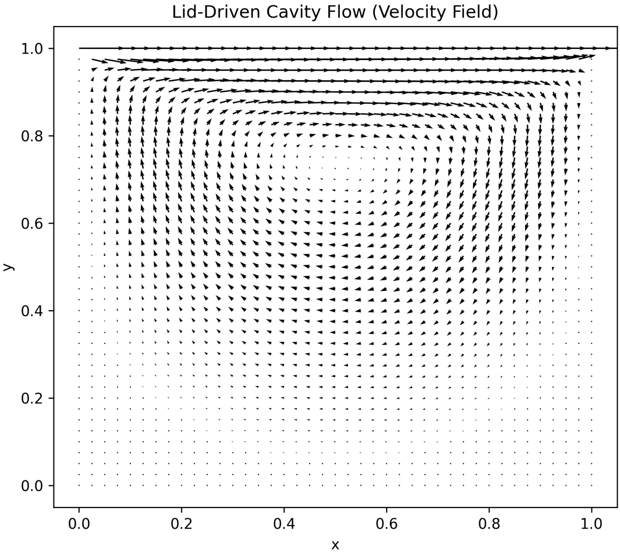
$$\frac{u_i^{n+1} - u_i^n}{\Delta t} + u_i^n \frac{u_{i+1}^n - u_{i-1}^n}{2\Delta x} + v_j^n \frac{u_{j+1}^n - u_{j-1}^n}{2\Delta y} = -\frac{\partial p}{\partial x} + \nu \nabla^2 u$$

- Pressure correction via SIMPLE or PISO algorithm

Boundary Conditions

- Top lid: $u = U_{lid}, \quad v = 0$
- Side and bottom walls: $u = v = 0$ (no-slip)
- Pressure: $\partial p/\partial n = 0$ at walls

Simulation Results



Observations

- Formation of primary vortex in cavity center
- Secondary corner vortices appear at higher Reynolds numbers
- Steady-state solution obtained after sufficient time steps
- Benchmark for testing CFD solvers and pressure-correction algorithms

Conclusion

- Lid-driven cavity flow is a standard CFD benchmark problem
- Numerical solution demonstrates incompressible flow behavior and vortex formation
- Useful for validating CFD methods like SIMPLE, PISO, or FVM solvers