

Numerical Algorithms

Esteban, Marie Joe, Amir, Kristin

May 5, 2025

Abstract

This document presents numerical algorithms and methods for solving 2D incompressible Navier-Stokes, with detailed sections on right- and left-hand side formulations, Poisson solver, and validation techniques.

1 Model

We are dealing with the 2D incompressible Navier-Stokes equations, which gives us:

- For the x-momentum equation:

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

- For the y-momentum equation:

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

- Additionally, the continuity equation for incompressible flow ensures mass conservation:

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

2 Method

We separate the computation into spatial discretization, time advancement of velocity (predictor), and pressure correction (corrector) to handle the incompressibility constraint. This method decouples the velocity and pressure solutions, solving the momentum equations first to get a tentative velocity (ignoring incompressibility initially), then correcting it by projecting the velocity field onto a divergence-free space via a pressure Poisson equation.

1. Spatial discretization strategies for the right-hand side of the momentum equations. This step involves discretizing the spatial terms (RHS) of the momentum equations, which includes the advection (convection), diffusion, and pressure gradient terms.
2. Then we handle the time derivative (left-hand side) of the momentum equations by simply taking the spatially discretized equations and advancing the velocity fields (u and v) in time without considering the continuity constraint ($\nabla u = 0$). This is the predictor step, computing a tentative velocity field based on the current state, which performs a transport phase that accounts for advection and diffusion without satisfying incompressibility.
3. Finally a new pressure field is computed to enforce incompressibility by solving a Poisson equation for pressure (relating pressure at any point to the entire velocity field). Then the velocity field is corrected to satisfy the continuity equation.

This method was preferred to a coupled (monolithic) method that solves the velocity and pressure fields simultaneously in a fully coupled system at each time step, since it decreases computational cost, although it cannot achieve higher-order temporal accuracy as easily. This projection method also makes it easier to analyze global error and convergence rate, and separate the work in different tasks.

3 Spatial Discretization

3.1 Finite Difference

3.1.1 Central Scheme

3.1.2 Upwind Scheme

3.2 Finite Volume

4 Prediction Step

4.1 Explicit Schemes

4.2 Semi-implicit Schemes

4.2.1 Explicit Advection and Implicit Diffusion

4.2.2 BDF with Extrapolation for Advection

5 Projection Step

6 Validation

6.1 Taylor-Green Vortex

The Taylor-Green Vortex (TGV) is a canonical benchmark problem for validating numerical solvers of the incompressible Navier-Stokes equations due to its exact analytical solution. This solution describes a decaying vortex flow in a periodic domain, allowing for direct comparison between simulated and analytical results, given by:

$$\begin{aligned}u(x, y, t) &= \cos(x) \sin(y) \exp(-2\nu t \pi^2), \\v(x, y, t) &= -\cos(y) \sin(x) \exp(-2\nu t \pi^2), \\p(x, y, t) &= \frac{1}{4} (\cos(2x) + \cos(2y)) \exp(-4\nu t \pi^2),\end{aligned}$$

where u and v are the velocity components in the x - and y -directions, respectively, p is the pressure, ν is the kinematic viscosity, and t is the time. The factor π^2 in the exponential decay term accounts for the specific domain scaling used in the implementation.

Validation is performed by computing the kinetic energy of the simulated flow and comparing it with the analytical kinetic energy at a given time, to obtain a quantitative measure of the solver's accuracy. Periodic boundary conditions are applied to maintain the flow's continuity across the domain boundaries.

6.2 Lid-Driven Cavity

The Lid-Driven Cavity flow is another benchmark for validating numerical solvers, particularly for assessing the handling of boundary conditions and steady-state flow features in a confined domain. Unlike the Taylor-Green Vortex, it lacks an analytical solution, so validation relies on comparison with established reference data from literature or prior simulations.

The problem setup involves a square domain of unit length with no-slip boundary conditions on three walls ($u = v = 0$) and a moving top wall (lid) with a constant tangential velocity ($u = 1, v = 0$). The flow is driven by the lid motion, resulting in a primary recirculating vortex and, at higher Reynolds numbers, secondary vortices in the corners. We use Neumann boundary conditions for pressure ($\frac{\partial p}{\partial n} = 0$) at the walls.

Here, validation focuses on the stream function's minimum value, which is indicative of the primary vortex strength in the cavity. According to Logg et al. (2012) this value after 2.5s is -0.061076605 . The comparison with a reference value allows for an assessment of numerical accuracy and convergence towards expected flow behavior.