MAE250: Project Overview and Checkpoints

(K. Taira, UCLA, Spring 2025)

The goal of this quarter-long course project is to develop a fully validated computer program that can simulate incompressible flow. The checkpoints that are assigned every two weeks provide milestones of the project which are to be put together as a complete CFD computer program for the final report. As an application, this course focuses on simulating a lid-driven cavity flow a schematic of which is shown in Fig. 1. This code could be further modified to simulate various flows by accordingly changing the boundary conditions.

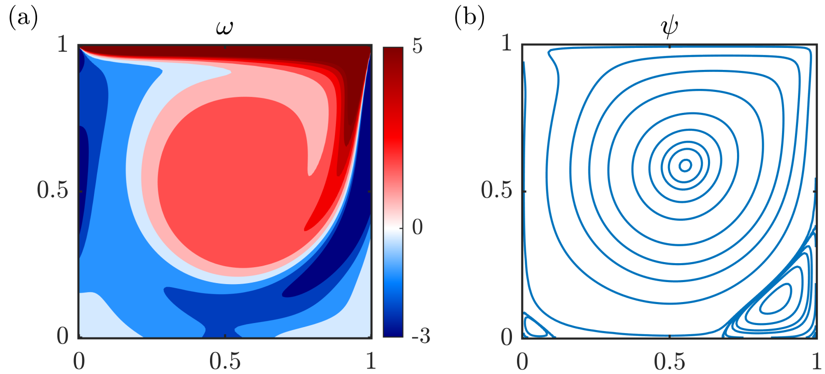

Figure 1: Illustration of (a) vorticity field (ω) and (b) streamfunction (ψ) contours for flow in a lid-driven cavity at $Re = 1000$.

Instructions

The report must be compiled using LaTeX in arXiv format[1]. We recommend the use of Overleaf for compiling your report. Attach your code along with your report (as a single document in PDF format). There is no limitation to the number of pages for the report. You will receive extra credit if you report results beyond what is requested in the problem statements. Some parts of the problems are meant to be open-ended to some degree. Your work must reflect your own understanding.

Checkpoint 1

Objectives: Verify the accuracy of different finite difference schemes and time-stepping schemes on the advection equation and generation of the staggered grid.

Verification of finite difference and time stepping methods

Numerically solve the advection equation

$$\frac{\partial f}{\partial t} + c\frac{\partial f}{\partial x} = 0 \tag{1}$$

with $c = 1$ over a periodic domain $x \in [0, 1]$. Follow the example shown in Figure 2.13 of the textbook and replicate the error behaviors discussed in class by using central differencing and upwind differencing. Try different time-stepping methods and report the error behavior based on the size of the time step. Discuss how you can reduce the errors.

[1] `https://www.overleaf.com/latex/templates/style-and-template-for-preprints-arxiv-bio-arxiv/fxsnsrzpnvwc`

Generation of staggered grid

Develop a code to define a two-dimensional rectangular computational domain as a staggered grid (see Fig. 2). The code should have the following parameters

- Computational domain $[0, L_x] \times [0, L_y]$.
- The domain is to be discretized with n_x and n_y cells in the x and y directions, respectively.
- The spatial domain should be discretized using a staggered grid (see Fig. 3.4 of [1]).
- Develop a routine to initiate your discrete velocity (u and v) and pressure (p) variables of appropriate sizes. Make sure to stack them into a vector format.
- Develop a routine to visualize velocity components (u and v), pressure (p), and vorticity (ω) over D.

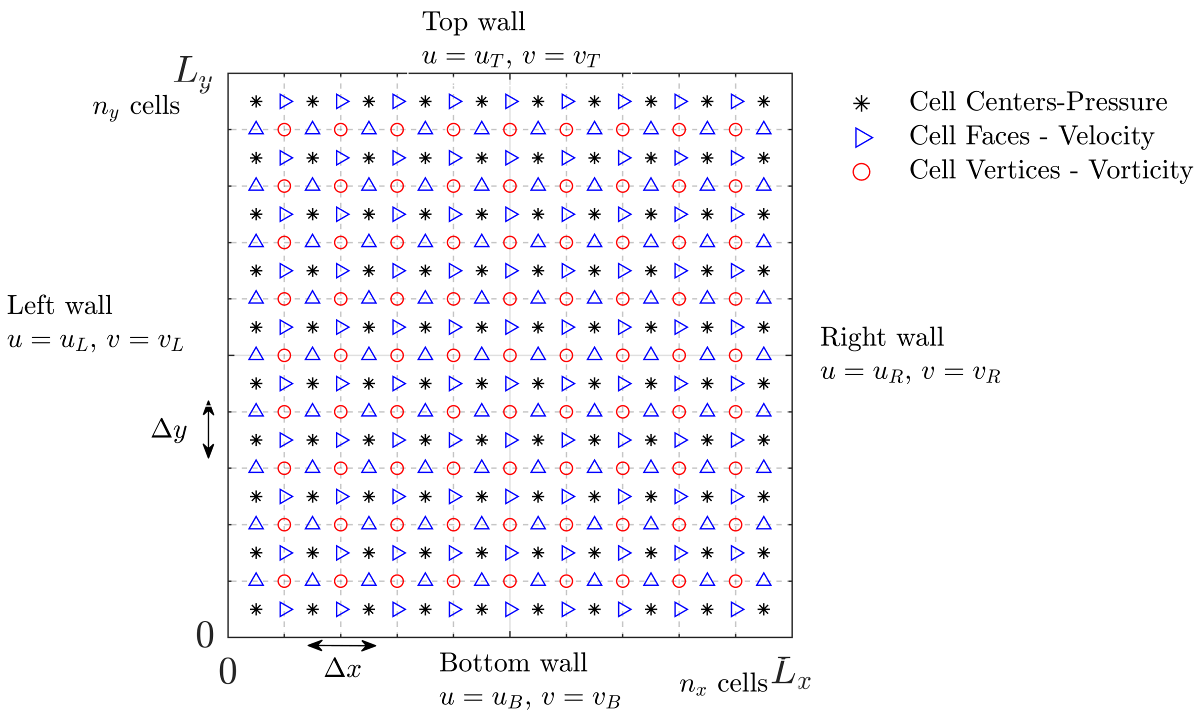

Figure 2: Schematic of staggered grid

Checkpoint 2/Midterm Report (Due: 11:59 pm, May 13, 2025)

Objectives: Verification of the various operators that appear in the Navier–Stokes equations and development of a conjugate gradient solver.

Verification of discrete operators

Develop routines that will perform the following discrete operations. You will need to perform verification for each of the operators below (this must be included in your report).

- Divergence ($D : n_q \to n_p$)
- Gradient ($G : n_p \to n_q$)

- Laplacian ($L : n_q \to n_q$) on velocity for viscous diffusion.
- Nonlinear advection ($N : n_q \to n_q$) for advection.

For each operator, perform a spatial convergence check and verify the associated order of the chosen finite differencing scheme using an analytical function as the input.

Verification of conjugate gradient solver

Develop a routine using a conjugate gradient solver to iteratively solve $Ax = b$

Checkpoint 3

Objectives: Validation of the code for a lid-driven cavity flow. Using the validated components of the incompressible flow solver, validate your results for a lid-driven cavity flow by comparing them with Ghia *et al.* [2]. The flow in a lid-driven cavity is simulated in a of domain $[0, 1] \times [0, 1]$ with the boundary condition for velocity $u_T = 1$ and rest all three walls are stationary.

The following items must be included in the report:

- Spatial and temporal convergence check of the solver
- Comparison of your simulated results with those from Ghia *et al.* at $Re = 400$ and 1000.
- Verification of streamlines with the corresponding contours (see Figure 3).
- Verification of velocity profiles (see Figures 2a and 2b)
- Verification of vorticity contours (see Figure 4).

Checkpoint 4/Final Report (Due: 11:59 pm, June 6, 2025)

Objectives: Modification of the solver to include a circular cylinder inside the lid-driven cavity using immersed-boundary method.

The surface of cylinder could be generated using a set of body forces added to the right hand side of the Navier–Stokes equations using the immersed-boundary projection method.

The following items must be included in the report:

- Verification of generation of cylinder surface.
- Verification of spatial accuracy of your scheme.

Final Report Requirements

As the overall project, you now have developed a computed program to simulate incompressible flow in a lid-driven cavity with an immersed body. This final report builds on your past reports and hence, if you have any missing items from earlier checkpoints, make sure to include them in the final report. Consider the final report to be a compilation of all past reports and efforts.

The following items must be included in the final report:

- Description of the overall approach to numerically solve for the incompressible flow.
- Verification of each operator used in the code.
- Spatial and temporal convergence check.
- Testing of the conjugate gradient solver.

- Validation for lid-driven cavity flow.
- Modification of the code to include a cylinder inside the cavity.
- Further modifications could be done as to either simulate a freestream flow or a channel flow by modifying the boundary conditions along the other walls. In addition, you can also perform any of the advanced analysis such as modal analysis for advanced characterization of the flow physics.

References

[1] T. Kajishima and K. Taira. *Computational fluid dynamics: incompressible turbulent flows*. Springer, 2016.

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