Universitat Politècnica de Catalunya

ESCUELA SUPERIOR DE INGENIERÍA INDUSTRIAL, AEROESPACIAL Y AUDIOVISUAL DE TERRASSA (ESEIAAT)

ASSIGNMENT 3: NUMERICAL RESOLUTION OF THE NAVIER-STOKES EQUATIONS

Computational Engineering - Space and Aeronautical Engineering MSc

Author:

Jorge SIMÓN AZNAR

Contents

1 Problem Specification

The Fractional Step Method is employed in the resolution of the Navier-Stokes equations within the context of the Lid-Driven Cavity problem. Fig 1.1 visually illustrates this scenario, where it's crucial to note that the top lid undergoes constant velocity motion. Despite the lid's movement, we presume constant mass within the system, as the mass entering the domain is considered equivalent to the mass exiting.

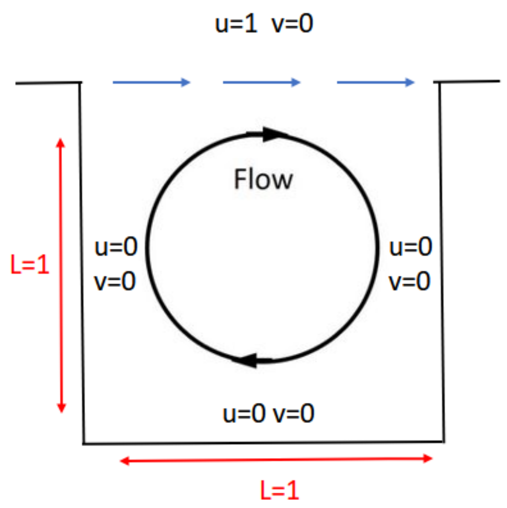

Figure 1.1: Lid-driven cavity.

To face this problem, it is crucial to take the Reynolds number into account. The formula used for its calculation is depicted in eq 1.1 below:

$$Re = \frac{UL}{\mu} \tag{1.1}$$

Here, μ represent the dynamic viscosity of the working fluid, while L and U stand for the characteristic length and velocity, respectively. It is important to note that in this analysis, fluid pressure is considered constant across all nodes within the domain. Consequently, any alteration in pressure at a single node will influence the pressures at all other nodes.

2 Introduction

In this study, we aim to address the Navier-Stokes (N-S) equations for a lid-driven cavity, a 2D fluid dynamics problem, employing the Fractional Step Method (FSM). Recognized for its superior performance and code simplicity, FSM involves solving the N-S equations in a series of fractional steps.

To initiate our discussion, let's present the N-S equations:

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}\nabla)\mathbf{u} = \nu \nabla^2 \mathbf{u} - \nabla p \tag{2.1}$$

$$\nabla \mathbf{u} = 0 \tag{2.2}$$

Here, $\mathbf{u}$ signifies the velocity, ν is the viscosity, and p represents the pressure. Equation (2.1) embodies the momentum equation, three equations in essence due to the three spatial components of velocity. Additionally, a third equation concerning energy conservation should be included, but here we focus on the mass conservation, given by Eq. (2.2).

The FSM technique builds upon the Helmholtz-Hodge (H-H) decomposition, stating that a vector field z within a bounded domain Ω can be expressed as the sum of a gradient field and a divergence-free vector parallel to the boundary:

$$\mathbf{z} = \mathbf{a} + \nabla \phi \;\; ; \;\; \nabla \mathbf{a} = 0 \;\; \mathbf{a} \in \Omega \tag{2.3}$$

This decomposition implies that the solutions of the N-S equations reside in the space of velocity fields.

To implement FSM for solving the N-S equations, we proceed as follows. We begin by rewriting the momentum equation (2.1) as:

$$\frac{\partial \mathbf{u}}{\partial t} = \mathbf{R}(\mathbf{u}) - \nabla p \tag{2.4}$$

where $\mathbf{R}(\mathbf{u}) \equiv -(\mathbf{u}\nabla)\mathbf{u} + \nu \nabla^2 \mathbf{u}$. We then perform a semi-discretization of the modified momentum equation and mass conservation (2.4 and (2.2), respectively):

$$\frac{\mathbf{u}^{n+1} - \mathbf{u}^{n}}{\Delta t} = \frac{3}{2}\mathbf{R}(\mathbf{u}^{n}) - \frac{1}{2}\mathbf{R}(\mathbf{u}^{n-1}) - \nabla p^{n+1} \tag{2.5}$$

$$\nabla \mathbf{u}^{n+1} = 0. \tag{2.6}$$

A predictor velocity, $\mathbf{u}^{p}$, is then computed to facilitate the calculation of the velocity in the next time step:

$$\frac{\mathbf{u}^{p} - \mathbf{u}^{n}}{\Delta t} = \frac{3}{2}\mathbf{R}(\mathbf{u}^{n}) - \frac{1}{2}\mathbf{R}(\mathbf{u}^{n-1}). \tag{2.7}$$

Applying the Helmholtz-Hodge theorem, it follows that:

$$\mathbf{u}^{\mathbf{n+1}} = \mathbf{u}^{p} - \Delta t \nabla p^{n+1} \tag{2.8}$$

and solving the Poisson equation (2.9) for p^{n+1}:

$$\nabla^2 p^{n+1} = \frac{1}{\Delta t} \nabla \mathbf{u}^p. \tag{2.9}$$

Finally, the velocity for the next time step is computed using the obtained expression:

$$\mathbf{u^{n+1}} = \mathbf{u}^p - \Delta t \nabla p^{n+1} \tag{2.10}$$

In summary, the FSM algorithm involves computing the predictor velocity $\mathbf{u}^p$, solving the Poisson equation for p^{n+1}, and calculating the velocity correction for the next time step. This iterative process continues until a steady state is reached. The algorithm requires both time and space discretization, typically employing a staggered grid, although further details about the grid concept are not explored in this work.

3 Code Structure

In this section, we outline the structural framework essential for solving the Navier-Stokes equations using the Fractional Step Method (FSM). The following steps delineate the key components of the code:

1. **Input Data**: Segregate input data into two sections: one focusing on the physical aspects, encompassing parameters like cavity dimensions (L and H), inlet velocity (u), initial thermodynamic variables—temperature, pressure, and density (T_{in}, P_{in}, ρ_{in}), Reynolds number, etc. The second section pertains to numerical aspects, including the quantity of control volumes (N and M), convergence criteria (δ), etc.

2. **Mesh Generation and Initial Conditions**: Generate the mesh and compute initial conditions for u^n, v^n, and p^n at each node.

3. **Velocity Predictor**: Compute the velocity predictor $\mathbf{u}^p$ by isolating it from eq. 2.7.

4. **Poisson Equation Solution**: Solve the Poisson equation, eq. 2.9, to obtain p^{n+1}.

5. **Convergence**: Check the convergence criteria: if $|p^{n+1} - p^n| < \delta$ is fulfilled, proceed to step 6. Otherwise, return to step 4, using the previous pressure solution as the new guess.

6. **Time Step**: Compute a new time step: $t^{n+1} = t^n + \Delta t$.

7. **Velocity Field**: Compute the velocity field for $n+1$ using eq. 2.10.

8. **Steady State Check**: if $|\mathbf{u}^{n+1} - \mathbf{u}^n| < \delta$ is fulfilled, proceed to step 9. Otherwise, set $\mathbf{u}^n = \mathbf{u}^{n+1}$ and return to step 3.

9. **Final Calculations and Output**: Conduct final calculations and print results.

This algorithm necessitates both time and space discretization, with the convergence of the Poisson equation and the velocity field serving as pivotal criteria for progression.

4 Expected Results

When solving the CFD challenge posed by a lid-driven cavity, the anticipated outcomes are notably shaped by the Reynolds number, a pivotal metric capturing the ratio of convective to diffusive terms. At lower Reynolds numbers, an expectation arises for a substantial recirculation zone within the cavity, often manifesting as two smaller vortices situated in the bottom corners [1]. With an escalation in the Reynolds number, these smaller vortices tend to intensify, and an additional vortex may materialize in the upper-left region of the cavity. In scenarios featuring very high Reynolds numbers, a common observation involves the emergence of two distinct recirculation zones or vortices, prominently visible in the lower-right corner Fig. 4.1.

The genesis of these vortices is attributed to internal friction within the fluid, a phenomenon intricately linked to the flow velocity. The specific behavior of the flow, encompassing characteristics such as vortex size and intensity, emanates from the delicate equilibrium between inertial and viscous forces within the system—a balance distinctly elucidated by the Reynolds number.

The figures presented below provide a visual representation of the solutions corresponding to different Reynolds numbers, encapsulating the diverse and intricate flow patterns inherent in the lid-driven cavity problem.

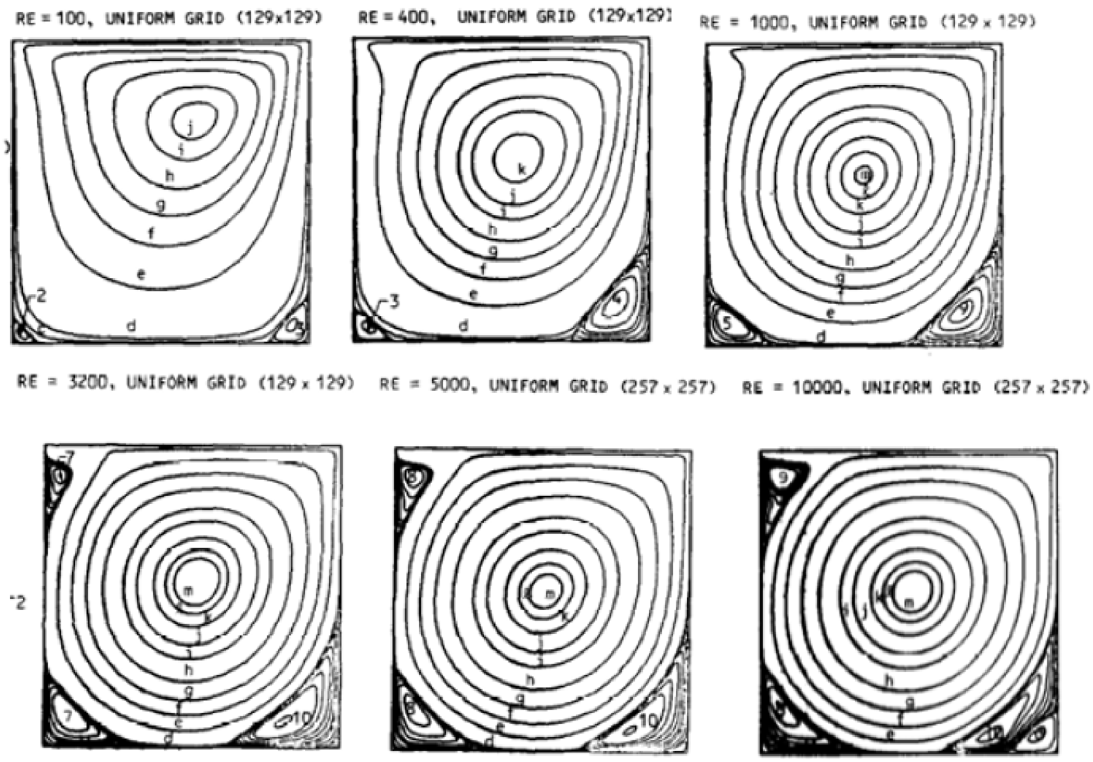

Figure 4.1: Solutions for $Re = 100, 400, 1000, 3200, 5000, 10000$ with a mesh of $129x129$ and $257x257$.

Bibliography

[1] U. Ghia, K. N. Ghia, and C. Shin, "High-re solutions for incompressible flow using the navier-stokes equations and a multigrid method", Journal of computational physics **48**, 387–411 (1982).