ME 5101
Computational Fluid Dynamics
Project

Part 1

February 24, 2019

1 Introduction

In this project, we will analyze the unsteady, viscous, incompressible, isothermal, two-dimensional, laminar flow of Newtonian fluid in a cavity covered with a lid. Consider a rectangular cavity ABCD of dimensions shown in Figure 1.1. AB, CD, and AD are rigid walls, whereas BC is open.

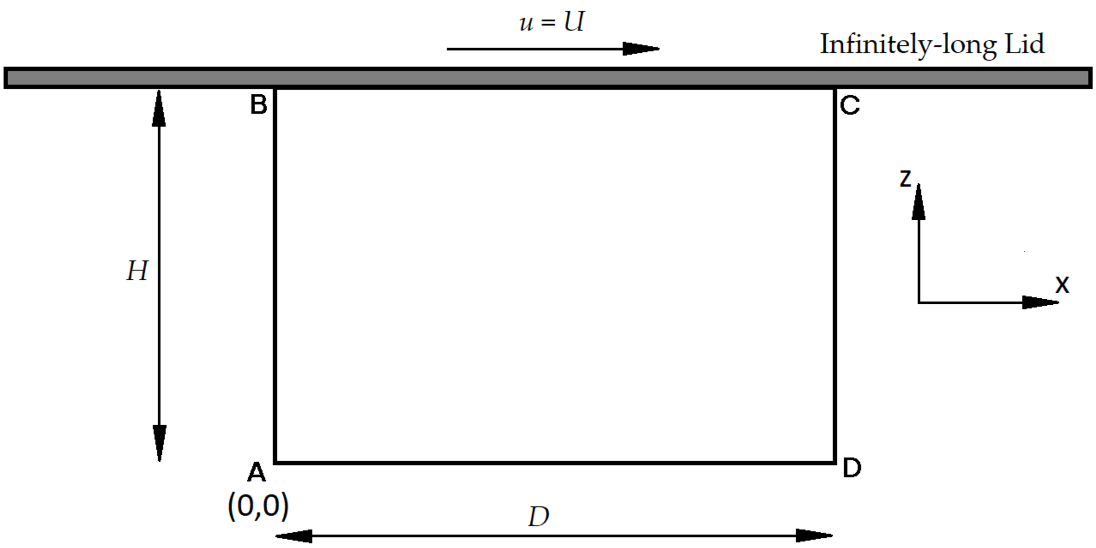

Figure 1.1: Geometry

The cavity if of length D and width H, in the x-z plane, as shown in Figure 1.1. The aspect ratio is defined as $R = H/D$. The Reynolds number is defined based on the velocity scale U and the length scale D. Acceleration due to gravity acts in the negative-y direction. The top of the cavity, BC, is covered with an infinitely-long rigid lid. Initially ($t \le 0$), the fluid inside the cavity is at rest. At time $t > 0$, the lid is set in motion to the right with a constant velocity U. You are to analyze the fluid flow inside the cavity for various conditions using proper governing equations, boundary conditions, initial conditions, and numerical schemes. Do a literature survey to see what this problem is all about!

2 Governing Equations

The governing equations are the equation of continuity and the Navier-Stokes equations (NSE), simplified for the conditions listed in the introduction. We will use a derived-variable approach involving the streamfunction and vorticiy. Derive these equations starting from the NSE. Discuss the advantages of using the streamfunction-vorticity method.

3 Initial and Boundary Conditions

Define appropriate initial and boundary conditions for this problem. The intial and boundary conditions will be for the velocity components. You must then derive appropriate initial and boundary conditions for the streamfunction and vorticity.

4 Nondimensionalization

Now, you must identify an appropriate length scale, velocity scale, and time scale for nondimensionalization. The scales for nondimensionalizing the streamfunction and vorticity equations should be derived from the basic length, velocity, and time scales. Using the derived scales, you must nondimensionalize the streamfunciton-vorticity equations and the initial/boundary conditions.

5 Discretization Schemes

We will first use the usual FTCS scheme and a grid that is not equally spaced (see Figure 5.1). Derive the orders of accuracies of this scheme for both time and space. The values of Δx and Δy are not constant. You must use an appropriate stretching function so that the number of grid points vary in such a way that more grid points are concentrated closer to walls and the lid. Why is this necessary? Comment on this!

Do a Von-Newmann stability analysis to find out the CFL criterion and the maximum time step to be used in your calculations. You may notice that $\{\Delta x,\ \Delta y,\ u,\ w\}$ are going to vary throughout the domain. So, the grid Courant number is also going to vary for every node, and every time step. So your code/program/algorithm must be adaptive; i.e, it must be able to choose the correct values of Δt so that the CFL criterion is satisfied at all times and at all locations in the domain.

Obtain a set of discretized equations consistent with the conditions mentioned above.

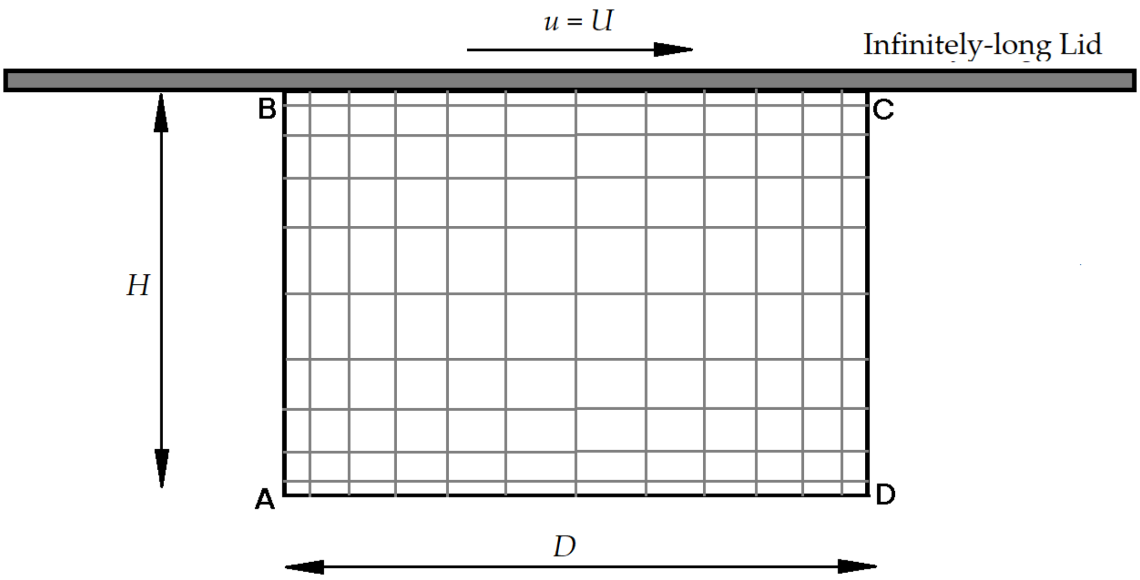

Figure 5.1: Grid

6 Algorithm

Now you have a nice system of discretized equations, and initial/boundary conditions. You must identify/work out a suitable procedure to solve these equations. This problem involves time marching. So you must start at $t = 0$ and march forward in steps using appropriate values of Δt. Note that Δt will not be constant here due to the CFL condition. So, you may not march in time in a linear fashion. For example, your time marching may be like this: $t = 0$, $t = 0.001$, $t = 0.0015$, $t = 0.003$, $t = 0.0032$, $t = 0.004$, ... and so on. So, your algorithm should be able to adapt to this nonlinear marching based on the CFL condition. Within each step, you must make sure that the solution is converged.

Use this algorithm to obtain the results asked for in §7. Remember to make your code as generic as possible, because subsequent parts of the project will involve modification of this code. Also write your code in a professional manner, and include comments whereever possible!

7 Results

7.1 Grid Independence Studies

Here, will use four grid sizes, viz., 128×128, 256×256, 512×512 and 1024×1024. Using these four grid sizes, find the following in the cavity at $t = 10$ s for $R = 1$ and $Re = 10$: (a) u vs z on the line $x = D/2$ and (b) w vs x on the line $z = H/2$. Create two figures - one for case (a) showing the curves using all four grid sizes in the same plot and another for case (b) showing the curves using all four grid sizes in the same plot. In each case, calculate the average value of u on $x = D/2$ and the average value of w on $z = H/2$. Tabulate these values for each grid size. Then, choose the minimum grid size for which these average values differ by less than 5% of their respective values for the next grid size. Use this grid size for all the simulations in §7.2.

7.2 Streamlines for Varying R and Re

1. Plot the streamlines for the flow at $t = 1$ s, $t = 5$ s and $t = 10$ s for $R = 1$ and $Re = 10$
2. Plot the streamlines for the flow at $t = 1$ s, $t = 5$ s and $t = 10$ s for $R = 1$ and $Re = 20$
3. Plot the streamlines for the flow at $t = 1$ s, $t = 5$ s and $t = 10$ s for $R = 1$ and $Re = 40$
4. Plot the streamlines for the flow at $t = 1$ s, $t = 5$ s and $t = 10$ s for $R = 1$ and $Re = 75$
5. Plot the streamlines for the flow at $t = 1$ s, $t = 5$ s and $t = 10$ s for $R = 2$ and $Re = 10$
6. Plot the streamlines for the flow at $t = 1$ s, $t = 5$ s and $t = 10$ s for $R = 2$ and $Re = 20$
7. Plot the streamlines for the flow at $t = 1$ s, $t = 5$ s and $t = 10$ s for $R = 2$ and $Re = 40$
8. Plot the streamlines for the flow at $t = 1$ s, $t = 5$ s and $t = 10$ s for $R = 2$ and $Re = 75$
9. Plot the streamlines for the flow at $t = 1$ s, $t = 5$ s and $t = 10$ s for $R = 0.5$ and $Re = 10$
10. Plot the streamlines for the flow at $t = 1$ s, $t = 5$ s and $t = 10$ s for $R = 0.5$ and $Re = 20$
11. Plot the streamlines for the flow at $t = 1$ s, $t = 5$ s and $t = 10$ s for $R = 0.5$ and $Re = 40$
12. Plot the streamlines for the flow at $t = 1$ s, $t = 5$ s and $t = 10$ s for $R = 0.5$ and $Re = 75$

8 Discussion

Comment on your results and discuss what you learned. In specific, comment on whether the FTCS scheme is appropriate for all the 12 cases you simulated.

This first part of the project will be due in the week of 12 March. I will let you know the exact date in a while. The entire project in this course will be done in stages (each stage or each part of the project will have a specific deadline, and the entire project will have a final deadline in the month of April), so stay tuned for the next part of the project! I am sending out this part now because some students approached me and informed that they are free during the semester break and they would like to start working on the project. So, ...

Please start working on this project during your semester break if time permits. Else, start atleast by the time you come back from the break. Please do not hesitate to meet me (please meet me in person, emails and phone calls won't be effective. If you are out of town during thr break, you could meet me after you're back!) if you have questions or if you get stuck in between. I will also be discussing this project in class in detial during the week of March 5.

At the end of the project, we will try to model heat transfer from electronic equipment using CFD. To reach this goal, I'd like to go in steps, so please cooperate with me. I am confident that the learning experience you gain while you work on this project will be useful to you!