ME 5101 Computational Fluid Dynamics Project

Report to be prepared in proper Manuscript Format

Complete report due before 6 PM on Mon, 18 March 19

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.

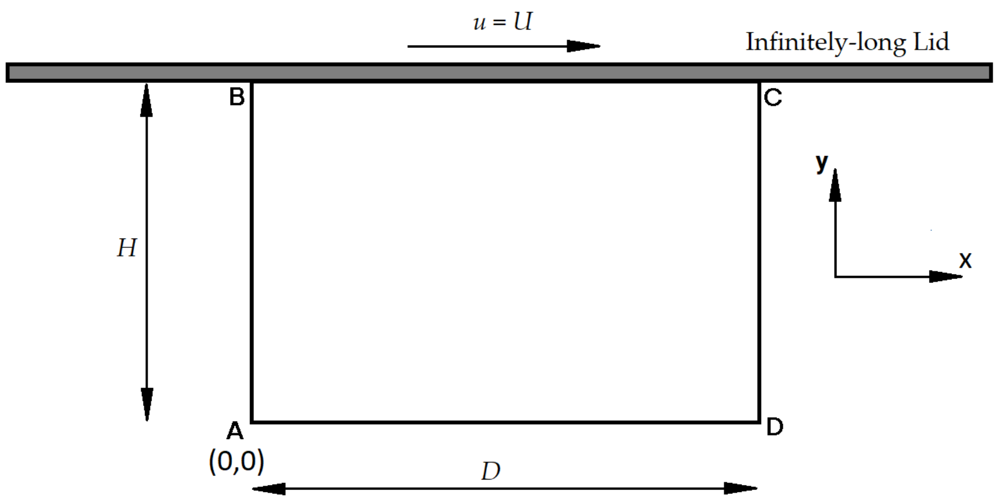

Figure 1.1: Geometry

The cavity is of length D and width H, in the x-y 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-z direction. The top of the cavity, BC, is covered with an infinitely-long rigid lid. Initially ($t \leq 0$), 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 appropriate length scales, velocity scales, and time scale for nondimensionalization. What are x_{ref}, y_{ref}, U_{ref}, V_{ref}?. 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 $\{\Delta x,\ \Delta y,\ u,\ v\}$ 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.
For §7.3, we will use a new scheme: the second-order upwind scheme. Learn what it is about!!

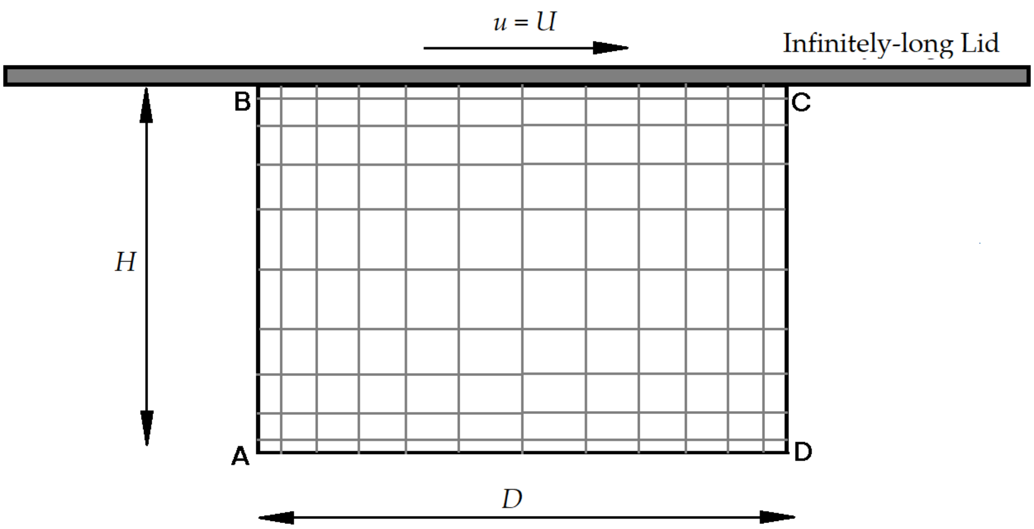

Figure 5.1: Grid

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 whereeever 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/U vs y/H on the line $x = D/2$ and (b) v/V_{ref} vs x/D 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/U on $x = D/2$ and the average value of v/V_{ref} on $y = H/2$. 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 $Re = 75$

7.3 Profiles for Steady Flow

Now, we will analyze the flow within the cavity at steady state. So, the time derivatives will dissappear in the governing equations. For this problem, you can use a uniform mesh. All discretization schemes that you use must be second order accurate. You must however, make sure that the

scheme is stable for all values of grid sizes $\{\Delta x, \Delta y\}$. To this end, we will use a second-order upwind scheme to discretize the convective terms. We will use two grids, 128 × 128 and 256 × 256 to simulate the flow within the cavity.
Plot the following for $AR = 1$ on the same graph and comment on your results:

1 Plot u/U (on X-axis) vs y/H (on Y-axis) on the vertical mid-plane ($x/D = 0.5$) for $Re = 100$ using the 128 × 128 grid.
2 Plot u/U (on X-axis) vs y/H (on Y-axis) on the vertical mid-plane ($x/D = 0.5$) for $Re = 100$ using the 256 × 256 grid.

Plot the following for $AR = 1$ on the same graph and comment on your results:

1 Plot v/V_{ref} (on Y-axis) vs x/D (on X-axis) on the horizontal mid-plane ($y/H = 0.5$) for $Re = 100$ using the 128 × 128 grid.
2 Plot v/V_{ref} (on Y-axis) vs x/D (on X-axis) on the horizontal mid-plane ($y/H = 0.5$) for $Re = 100$ using the 256 × 256 grid.

Plot the following in the same graph, for $AR = 1$ and using the 256 × 256 grid.

1 Plot u/U (on X-axis) vs y/H (on Y-axis) on the vertical mid-plane ($x/D = 0.5$) for $Re = 100$.
2 Plot u/U (on X-axis) vs y/H (on Y-axis) on the vertical mid-plane ($x/D = 0.5$) for $Re = 400$.
3 Plot u/U (on X-axis) vs y/H (on Y-axis) on the vertical mid-plane ($x/D = 0.5$) for $Re = 1000$.
4 Plot u/U (on X-axis) vs y/H (on Y-axis) on the vertical mid-plane ($x/D = 0.5$) for $Re = 3200$.
5 Plot u/U (on X-axis) vs y/H (on Y-axis) on the vertical mid-plane ($x/D = 0.5$) for $Re = 5000$.
6 Plot u/U (on X-axis) vs y/H (on Y-axis) on the vertical mid-plane ($x/D = 0.5$) for $Re = 7500$.

In the same graph, plot the benchmark values given below in Figure 7.1 using dots/squares/triangles etc. Comment on your results.

	Re						
y/H	100	400	1000	3200	5000	7500	10,000
1.00000	1.00000	1.00000	1.00000	1.00000	1.00000	1.00000	1.00000
0.9766	0.84123	0.75837	0.65928	0.53236	0.48223	0.47244	0.47221
0.9688	0.78871	0.68439	0.57492	0.48296	0.46120	0.47048	0.47783
0.9609	0.73722	0.61756	0.51117	0.46547	0.45992	0.47323	0.48070
0.9531	0.68717	0.55892	0.46604	0.46101	0.46036	0.47167	0.47804
0.8516	0.23151	0.29093	0.33304	0.34682	0.33556	0.34228	0.34635
0.7344	0.00332	0.16256	0.18719	0.19791	0.20087	0.20591	0.20673
0.6172	−0.13641	0.02135	0.05702	0.07156	0.08183	0.08342	0.08344
0.5000	−0.20581	−0.11477	−0.06080	−0.04272	−0.03039	−0.03800	0.03111
0.4531	−0.21090	−0.17119	−0.10648	−0.86636	−0.07404	−0.07503	−0.07540
0.2813	−0.15662	−0.32726	−0.27805	−0.24427	−0.22855	−0.23176	−0.23186
0.1719	−0.10150	−0.24299	−0.38289	−0.34323	−0.33050	−0.32393	−0.32709
0.1016	−0.06434	−0.14612	−0.29730	−0.41933	−0.40435	−0.38324	−0.38000
0.0703	−0.04775	−0.10338	−0.22220	−0.37827	−0.43643	−0.43025	−0.41657
0.0625	−0.04192	−0.09266	−0.20196	−0.35344	−0.42901	−0.43590	−0.42537
0.0547	−0.03717	−0.08186	−0.18109	−0.32407	−0.41165	−0.43154	−0.42735
0.0000	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000

Figure 7.1: Values of u/U vs y/H on ($x/D = 0.5$) for various Reynolds numbers.

Plot the following in the same graph, for $AR = 1$ and using the 256×256 grid.

1 Plot v/V_{ref} (on Y-axis) vs x/D (on X-axis) on the horizontal mid-plane ($y/H = 0.5$) for $Re =$ 100.
2 Plot v/V_{ref} (on Y-axis) vs x/D (on X-axis) on the horizontal mid-plane ($y/H = 0.5$) for $Re =$ 400.
3 Plot v/V_{ref} (on Y-axis) vs x/D (on X-axis) on the horizontal mid-plane ($y/H = 0.5$) for $Re =$ 1000.
4 Plot v/V_{ref} (on Y-axis) vs x/D (on X-axis) on the horizontal mid-plane ($y/H = 0.5$) for $Re =$ 3200.
5 Plot v/V_{ref} (on Y-axis) vs x/D (on X-axis) on the horizontal mid-plane ($y/H = 0.5$) for $Re =$ 5000.
6 Plot v/V_{ref} (on Y-axis) vs x/D (on X-axis) on the horizontal mid-plane ($y/H = 0.5$) for $Re =$ 7500.

In the same graph, plot the benchmark values given below in Figure 7.2 using dots/squares/triangles etc. Comment on your results.

	Re						
x/D	100	400	1000	3200	5000	7500	10,000
1.0000	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
0.9688	−0.05906	−0.12146	−0.21388	−0.39017	−0.49774	−0.53858	−0.54302
0.9609	−0.07391	−0.15663	−0.27669	−0.47425	−0.55069	−0.55216	−0.52987
0.9531	−0.08864	−0.19254	−0.33714	−0.52357	−0.55408	−0.52347	−0.49099
0.9453	−0.10313	−0.22847	−0.39188	−0.54053	−0.52876	−0.48590	−0.45863
0.9063	−0.16914	−0.23827	−0.51550	−0.44307	−0.41442	−0.41050	−0.41496
0.8594	−0.22445	−0.44993	−0.42665	−0.37401	−0.36214	−0.36213	−0.36737
0.8047	−0.24533	−0.38598	−0.31966	−0.31184	−0.30018	−0.30448	−0.30719
0.5000	0.05454	0.05186	0.02526	0.00999	0.00945	0.00824	0.00831
0.2344	0.17527	0.30174	0.32235	0.28188	0.27280	0.27348	0.27224
0.2266	0.17507	0.30203	0.33075	0.29030	0.28066	0.28117	0.28003
0.1563	0.16077	0.28124	0.37095	0.37119	0.35368	0.35060	0.35070
0.0938	0.12317	0.22965	0.32627	0.42768	0.42951	0.41824	0.41487
0.0781	0.10890	0.20920	0.30353	0.41906	0.43648	0.43564	0.43124
0.0703	0.10091	0.19713	0.29012	0.40917	0.43329	0.44030	0.43733
0.0625	0.09233	0.18360	0.27485	0.39560	0.42447	0.43979	0.43983
0.0000	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000

Figure 7.2: Values of v/U vs x/D on ($y/H = 0.5$) for various Reynolds numbers.

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 in §7.2. Also comment on the following for your simulations in §7.3: Is the 128×128 grid sufficient for simulations? Is the upwind scheme good enough? How do your values compare to the benchmark results given in Figures 7.1 and 7.2? Frame a couple of such questions of your own and answer them in this section!! Examples are: When you plot the stream line contours, do you see any primary and secondary vortices? What are the structures of these vortices for varying Re? How do the vorticity plots look like? etc.

This first part of the project is due before 6 PM on Monday, 18th March 2019. Please do all that I have asked you to do in this project, but it is not necessary that you stay restricted to this. Innovative, useful, and meaningful extra work will receive extra credit!!!

Please prepare the report using a word processor or LaTeX in standard manuscript format. If it gets cumbersome when you use a word processor/LaTeX, then you may submit a neatly handwritten report. I WILL NOT ACCEPT A HANDWRITTEN REPORT THAT WILL MAKE ME BREAK MY HEAD TO UNDERSTAND WHAT YOU HAVE WRITTEN.

Standard manuscript format means that you number and label all equations, figures, tables, titles, headings, subtitles etc., and that you use these numbers/labels when you refer to the respective item (equations, figures, tables, titles, headings, subtitles) in the description/text. In your project report, please include a detailed description of whatever I discussed in class today (06 Mar 19). You may add more subtiles as you deem fit to make things more clear.

Appendix

Attach your codes/programs!