Project 4 – Finite Volume Discrete Boltzmann Method (FVDBM)

Instructor: Dr. Leitao Chen

Assigned on: April 15, 2025

Due at: 11:59 PM on April 29, 2025

Notes in the front:

- Start working on the project as soon as possible.
- The project is individual. Group discussions are encouraged. But you cannot reply on your peers to complete the project.
- No handwritten report will be accepted. The report must be a well-formatted, well-organized, and typewritten document that includes all necessary work processes, including but not limited to
  - ✓ schematics that are used to illustrate the problems and methods;
  - ✓ executable MATLAB codes that are attached at the end of the report (codes can be attached as texts at the end of the report or submitted as separate files);
  - ✓ simulation results presented as figures, tables, charts, or any other format that is needed;
  - ✓ discussions and comments that accompany your work process, observations, results, questions, and concerns, etc.
- The late policy in the syllabus applies.
- Plagiarism check is strictly enforced. If the report or code is found copied from other sources, it receives a 0 score automatically.

Grading scheme:

The project report will be graded in the following areas:

- **Completeness (25%)**
  Ask yourself: whether your report contains all the necessary components.
- **Correctness (25%)**
  Ask yourself: whether your methods, schematics, codes, analysis, results, and conclusions are correct.
- **Formality, organization, and readability (25%)**
  Ask yourself: whether your report is well-formatted, well-organized, well-written with no errors or typos, and fun to read.
- **MATLAB code (25%)**
  Ask yourself: whether your codes can generate the same results you showed in your project report, and whether your code is well-organized, well-commented, free of bugs, and easy to read.

Problem Statement (100 Points + 20 Bonus Points):

The lid-driven square cavity flow is defined in Fig. 1, in which L is the length of each side. The top boundary is moving with a speed of U pointing towards the positive x-direction. The top boundary is therefore called the moving lid. All other boundaries are stationary with non-slip boundary conditions. At a steady state, the fluid will rotate clockwise inside the cavity. This is also a very important benchmark problem since it is highly nonlinear, caused by the singularities at the two top corners of the moving lid. The flow is characterized by *Re* of the flow, which is defined as

$$Re = \frac{UL}{\upsilon} \tag{1}$$

where υ is the kinematic viscosity of the fluid.

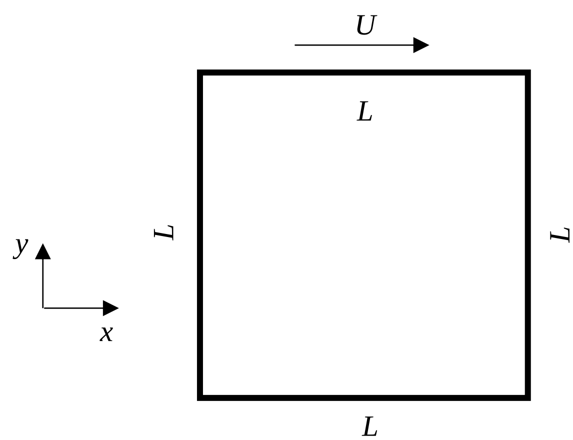

Figure 1. Lid-driven square cavity flow

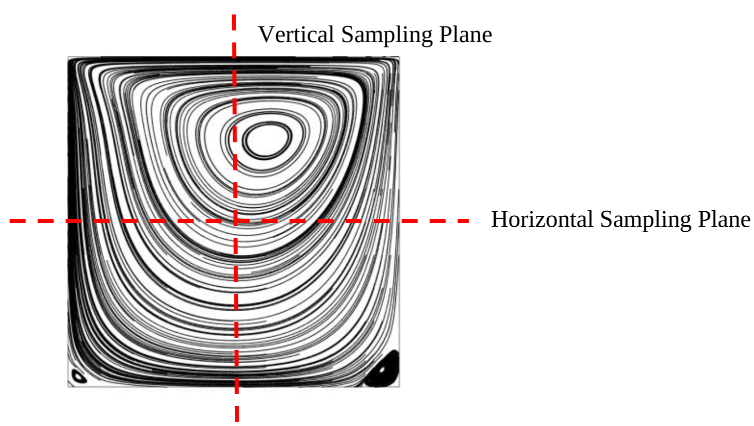

Figure 2. Solution of lid-driven square cavity flow at *Re*=100

Unfortunately, there is no analytical solution for the lid-driven square cavity flow. Therefore, the benchmark data is usually from experiments and numerical solutions. A very popular one is from Ghia [1], in which a numerical solution is generated by solving the Navier-Stokes equation. The solution from [1] is provided here. However, lid-driven cavity flow is a very common problem and there are numerous publications. You are encouraged to use benchmark data from other sources too.

At Re=100, the streamline of the steady-state solution is shown in Fig. 2. The solution from Ghia [1] is sampled on the horizontal and vertical mid-plane slices, as shown in Fig. 2 as well. On the horizontal sampling plane, the benchmark data is provided in Tab. 1, in which y* and u* are normalized y-coordinate and velocity in the x-direction, respectively, as defined in Eq. (2) and Eq. (3), in which y and u are the physical y-coordinate and physical velocity in the x-direction.

$$y^* = \frac{y}{L} \tag{2}$$

$$u^* = \frac{u}{U} \tag{3}$$

$y*$	$u*$
0	0
0.0547	-0.03717
0.0625	-0.04192
0.0703	-0.04775
0.1016	-0.06434
0.1719	-0.1015
0.2813	-0.15662
0.4531	-0.2109
0.5	-0.20581
0.6172	-0.13641
0.7344	0.00332
0.8516	0.23151
0.9531	0.68717
0.9609	0.73722
0.9688	0.78871
0.9766	0.84123
1	1

Table 1. Benchmark data on the horizontal sampling plane at *Re*=100

On the vertical sampling plane, the benchmark data is provided in Tab. 2, in which x^* and v^* are normalized x-coordinate and velocity in the y-direction, respectively, as defined in Eq. (4) and Eq. (5), in which x and v are the physical x-coordinate and physical velocity in the y-direction.

$$x^* = \frac{x}{L} \tag{4}$$

$$v^* = \frac{v}{U} \tag{5}$$

x^*	v^*
0	0
0.0625	0.09233
0.0703	0.10091
0.0781	0.1089
0.0938	0.12317
0.1563	0.16077
0.2266	0.17507
0.2344	0.17527
0.5	0.05454
0.8047	-0.24533
0.8594	-0.22445
0.9063	-0.16914
0.9453	-0.10313
0.9531	-0.08864
0.9609	-0.07391
0.9688	-0.05906
1	0

Table 2. Benchmark data on the vertical sampling plane at *Re*=100

(a) (20 Points) Develop an FVDBM model for this problem based on **square control volumes** using **the 1st-order upwind scheme for the flux calculation**, a **fully explicit scheme (EXEX) for time marching**, and **the boundary scheme in the lecture for the boundary conditions**. Simulate the lid-driven cavity flow at Re=100 and obtain the steady-state solution. Plot the **normalized** results on the vertical sampling plane (y^* vs. u^*) and on the horizontal sampling plane (v^* vs. x^*) and compare them with Ghia's results

[1] in Tab. 1 and Tab. 2 respectively. Use discrete markers for Ghia's results and use lines for your simulation results. An example figure is shown below in Fig. 3. Discuss your observations by focusing on the errors between your results and Ghia's results (consider to quantify the errors).

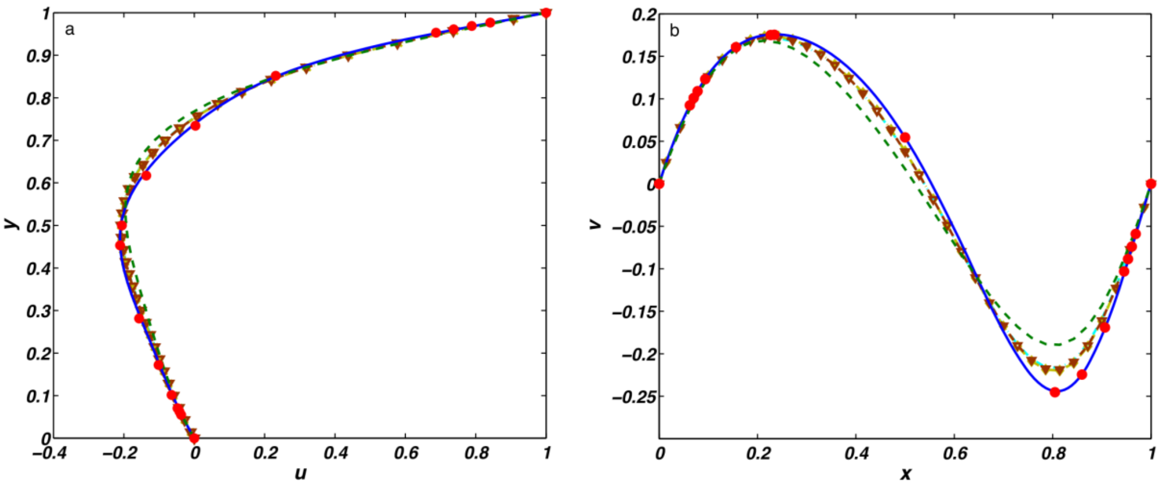

Figure 3. Example of flow velocities of lid-driven square cavity flow at *Re*=100. The left plot is for y^* vs. u^* on the vertical sampling plane and the right plot is for v^* vs. x^* on the horizontal sampling plane. Red dots are Ghia's results and lines are numerical results.

(b) **(20 Points)** You may discover that your results are still quite off from Ghia's results. Given the computational resource you have, use the finest mesh you can afford. Then, repeat part (a) by using this refined mesh. Add your new results to the plots you have in part (a). Do you see improvement? How many control volumes do you have for the new results? How long does your simulation take to get the results?

(c) **(20 Points)** The source for the error is the diffusion error associated with the 1st-order upwind scheme. We can reduce this diffusion error by using higher-order flux scheme. Repeat part (a) by using the 2nd-order Lax-Wendroff scheme. Make sure the mesh resolutions are the same in part (a). Add your new results into the plots you have in part (b). Do you see improvements, compared to part (a) and part (b)? How much longer (or shorter) does your simulation take to get the results compared to part (a) and part (b)? Do you think it is worth to use the 2nd-order Lax-Wendroff scheme?

(d) **(20 Points)** Similar to what you did in part (b), use the finest mesh you can afford to re-run the model with the 2nd-order Lax-Wendroff scheme. Then, repeat part (c) by using this refined mesh. Add your new results into the plots you have in part (c). Do you see improvement, compared to part (a), (b) and (c)? How many control volumes do you have for the new results? How much longer (or shorter) does your simulation take to get the results compared to part (a), (b) and (c)? Can you have the results with a spot-on accuracy compared to Ghia's results, such as the blue lines in Fig. 3?

(e) (20 Points) Now, develop the implicit-collision-and-explicit-flux (IMEX) time-marching for your model. How is the stability of the new model compared to the one with the fully explicit (EXEX) time-marching? In order to have a good comparison, do the following things:

- **Step 1:** Pick a mesh size and keep it the same for EXEX and IMEX models. Use the same 1st-order upwind scheme for the flux calculation in both models.
- **Step 2:** Decrease τ, then keep it the same for EXEX and IMEX models. In order to know the error of the model, also decrease U in Eq. (1) to keep Re=100. By doing, you can always compare your results with Ghia's. Run both EXEX and IMEX models to see if the simulations explode. If not, keep decreasing τ value for EXEX and IMEX models and re-run the models and answer these questions:
    - ❖ At what τ value, the model with EXEX time-marching start to explode?
    - ❖ At what τ value, the model with IMEX time-marching start to explode?
    - ❖ Which time-marching scheme tend to explode when decreasing τ, EXEX or IMEX?
- Step 3: Repeat Step 1 and Step 2 for the model with the 2nd-order Lax-Wendroff scheme for flux calculations.

(f) (Bonus 20 Points) You may realize at this point that the FVDBM code is much slower than the LBM code. What might be the reasons causing this? If you try to optimize the code, what would you do? Please optimize the code to show it still produces the correct results. Then, benchmark the speeds before and after the optimization.

References

[1] U. Ghia, K. N. Ghia, and C. T. Shin, "High-re solutions for incompressible flow using the Navier-Stokes equations and multigrid method," J. Comput. Phys. 48, 387 (1982).