

PAPER • OPEN ACCESS

Modeling and Simulation of Top and Bottom Lid Driven Cavity using Lattice Boltzmann Method

To cite this article: K A Yuana *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **546** 052088

View the [article online](#) for updates and enhancements.



IOP | ebooks™

Bringing you innovative digital publishing with leading voices to create your essential collection of books in STEM research.

Start exploring the **collection** - download the first chapter of every title for free.

Modeling and Simulation of Top and Bottom Lid Driven Cavity using Lattice Boltzmann Method

K A Yuana^{1,2,*}, E P Budiana³, Deendarlianto¹ and Indarto¹

¹Mechanical Engineering Department, Faculty of Engineering, Gadjah Mada University, Bulaksumur, Sleman, Yogyakarta, Indonesia.

²Networking and Software Engineering Dept., Computer Science Faculty, Amikom University Yogyakarta, Ring Road Utara, Sleman, Yogyakarta, Indonesia.

³Mechanical Engineering Department, Faculty of Engineering, Sebelas Maret University, Jl. Ir. Sutami 36A, Jebres, Surakarta, Central Java, Indonesia

*Corresponding Author: arikumara1506@gmail.com

Abstract. Simulation of computational fluid dynamic (CFD) in isothermal fluid flow is very important in engineering and modeling problem. The simulation can explain the closed-flow behavior as complement with experimental field. Lid Driven Cavity as the tool for CFD benchmark is important to investigate. Lattice Boltzmann Method (LBM) as non-conventional CFD is used as present method investigation. The keys parameters of lid driven cavity problem are Reynolds number and the parts of moving boundaries that driven the cavity. In the present investigation was used the variations of Reynolds numbers: 500, 1000 and 2000. The lid that driven the cavity are top and bottom part. The results presented in visual flow pattern and in two dimensional velocity plot.

1. Introduction

Lid driven cavity (LDC) is the phenomena where closed flow happens only in momentum transport, that means there is neither heat transport nor species transport [1]. The phenomena of closed flow in industrial could be seen in chemical etching or film coating. The classic LDC problem has been investigated since 60's decade and developed very fast until current era as the development of computer technology also very vast. The computer technology as the prime mover for numerical technique development is unavoidable. The quite comprehensive review about classic LDC where only one part of Lid that move has done by some authors. Marchi et al. (2009) reviewed the 18 papers of classic LDC from 1961 until their own work time. Almost all of their works were based on Navier-Stokes governing equations and only one that used Lattice Boltzmann Method [2]. The most popular method to discretize the mathematics models in the conventional Navier-Stokes based are Finite Difference methods. The Reynolds number that were used were various from low to 10,000 and even one of them until 21,000. Abdel Migid et al. (2016) also reviewed 27 papers of classic LDC from 1966 until 2014 to be compared with their own work. Only two of them used LBM instead of Navier-Stokes based and Reynolds number varied from low to 10,000 and one of them until 15,000 [3].

The recent most interesting LDC variations is when two walls diametrically facing each other move in the same direction and same velocity, while the two others walls are kept still [1]. The interesting phenomena that need to be investigated are the structure of symmetric and non-symmetric flows due to



flow interactions. Lemée *et al.* (2015) reported the symmetric flows is stable and steady for Reynolds number under 4000. Conventional CFD has passed half century using LDC as benchmarking tools that based on Navier-Stokes governing equation. One quite comprehensive investigations about various technique in solving LDC problem did by AbdelMigid *et al.* (2016) [3]. Finite Difference method sit the most popular tool, then Finite Volume method sit the second place then being followed by Finite Element method in the third place.

Besides of the Navier-Stokes based computational fluid dynamics (CFD) as conventional simulation, Lattice Boltzmann Method (LBM) is emerged as promising direct numerical simulation tool. The main advantage of LBM over the Navier-Stokes CFD are: (1) more fundamental paradigm rather than continuum conventional approach due to build from mesoscopic kinetic molecular interactions (2) the pressure is calculated through equations of state (EOS) so that not necessarily solve Poisson equation (that computationally expensive) (3) LBM easy to parallelize due to the locality in computation and (4) the bounce back scheme that easy to handle to treat no slip boundary condition [4] [5].

In this paper the LDC was modeled and solved using LBM with top and bottom wall moved to the right with normalized velocity equal to one and Reynolds number 500, 1000 and 2000. Scheme of presentations are governing equations and modelling next to this part, then results and discussions and finally the conclusions.

2. Governing equations and modeling

The basic idea of Lattice Boltzmann method is the parameter that describes distribution functions of the collected particles in the fluid system and express in equation as [4] [6]:

$$f(r+c\Delta t, c+F\Delta t, t+\Delta t)drdc - f(r,c,t)drdc = \Omega(f)drdc\Delta t \quad (1)$$

where, $f(r,c,t)drdc$ is collected particles distribution function at space position r , velocity c at time t , before get the external force F . The next distribution function after being exposed to the external force become $f(r+c\Delta t, c+F\Delta t, t+\Delta t)drdc$ and the quantity difference of collected particles before and after collisions are as much as $\Omega(f)drdc\Delta t$. If equation (1) is divided by $drdc\Delta t$ and with $\Delta t \rightarrow 0$, then will be resulted:

$$\frac{df}{dt} = \Omega(f)$$

(2)

Equation (2) shows that the total difference of speed molecule distribution is equal to the total collision change. Consequently, as f is a function of r , c and t , then:

$$df = \frac{\partial f}{\partial r} dr + \frac{\partial f}{\partial c} dc + \frac{\partial f}{\partial t} dt,$$

and if both sides is divided by dt , result:

$$\frac{df}{dt} = \frac{\partial f}{\partial r} \frac{dr}{dt} + \frac{\partial f}{\partial c} \frac{dc}{dt} + \frac{\partial f}{\partial t} = \frac{\partial f}{\partial r} c + \frac{\partial f}{\partial c} a + \frac{\partial f}{\partial t},$$

where a is acceleration and $a=F/m$ (Newton's law II). Then equation (2) becomes:

$$\frac{df}{dt} + \frac{\partial f}{\partial r} c + \frac{\partial f}{\partial c} \frac{F}{m} = \Omega$$

(3)

Operator of Ω approached by Bhatnagar, Gross dan Krook (BGK) as:

$$\Omega = \omega(f^{eq} - f) = \frac{1}{\tau} (f^{eq} - f) \quad (4)$$

Where ω is coefficient of molecule collision frequency and expressed as $\omega = \frac{1}{\tau}$, and τ is relaxation factor.

Equation (3) without external force can be expressed as:

$$\frac{df}{dt} + c\nabla f = \frac{1}{\tau} (f^{eq} - f) \quad (5)$$

Discretization of equation (5) results :

$$\frac{df_i}{dt} + ci\nabla f_i = \frac{1}{\tau} (f_i^{eq} - f_i) \quad (6)$$

or:

$$f_i(r + c_i\Delta t, t + \Delta t) = f_i(r, t) + \frac{\Delta t}{\tau} [f_i^{eq}(r, t) - f_i(r, t)] \quad (7)$$

Macroscopic parameters such as density and velocity of fluid flow expressed as:

$$\rho = \sum_i f_i^{eq} \quad (8)$$

$$\rho c = \sum_i c_i f_i^{eq} \quad (9)$$

Scheme of path in LBM expressed as $DnQm$, where n shows dimension and m shows number of all possibilities velocity direction. This paper use D_2Q_9 means 2 dimension and 9 speed velocity direction as on Figure 1.

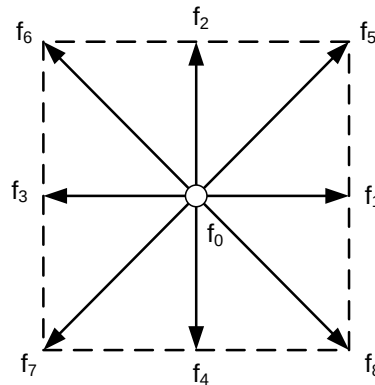


Figure 1. D_2Q_9 Scheme

For the model of D_2Q_9 the discrete velocities are given by:

$[e_0, e_1, e_2, e_3, e_4, e_5, e_6, e_7, e_8] = c \begin{bmatrix} 0 & 1 & 0 & -1 & 0 & 1 & -1 & -1 & 1 \\ 0 & 0 & 1 & 0 & -1 & 1 & 1 & -1 & -1 \end{bmatrix}$ where c is lattice speed that is defined as $c = \Delta x / \Delta t$, where Δx is discrete space unit and Δt is discrete time unit.

The scheme of Top and Bottom Lid Driven Cavity is shown on Figure 2. The condition is the square cavity with top lid and bottom lid moving with same direction and normalized velocity. The left and right wall are kept still. The boundary condition that appropriate for situation as shown in Figure 2 in LBM called bounce-back boundary condition.

The dimensionless Reynolds number that being used defined as [4]:

$$Re = \mathbf{u}L/\nu \quad (10)$$

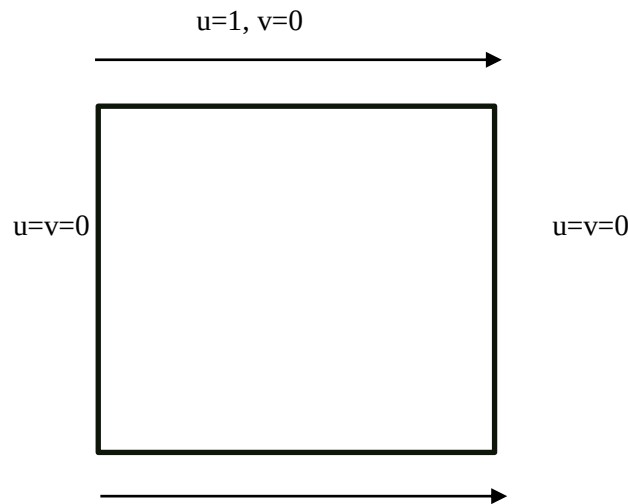


Figure 2. Scheme of Top and Bottom Lid Driven Cavity

where $\mathbf{u}$ is characteristic velocity, L is characteristic length and ν is fluid viscosity. The application of Reynolds number in lid driven case is defined as:

$$Re = (\text{lid driven velocity})(\text{lattice number})/(\text{viscosity}) \quad (11)$$

The lattice number in this present work are 100×100 .

3. Results and discussions

First part of these discussions is the validation of present work by benchmarking with the outstanding existing work. One of the prominent works that frequently used by researchers in CFD of LDC case is that worked by Ghia et al. [7]. Using conventional Navier-Stokes governing equations and was solved by multi-grid method, Ghia et al. present the result of single top lid driven cavity with some Reynolds number. Present work will validate using the Ghia's et al. work at Reynolds number of 1000 and plotted the velocities of both x and y directions in two-dimensions Cartesian coordinate system. The results are shown on Figure 3 (a) in velocity of x -direction (u) vs. lattice positions and Figure 3 (b) in velocity of y -direction (v) vs. lattice positions. All lattice positions are take in the mid cross-section of horizontal cavity. As it were shown on the both Figure 3(a) and Figure 3(b), the present work that using LBM is almost exactly similar with Ghia's work.

The results of double top and bottom LDC in present work are presented in each Reynolds number of 500, 1000 and 2000. The visual simulations are shown in stream functions and velocity quivers to show the flow directions. Finally, will be shown the velocity of x and y direction vs lattice and presented in all three Reynolds number.

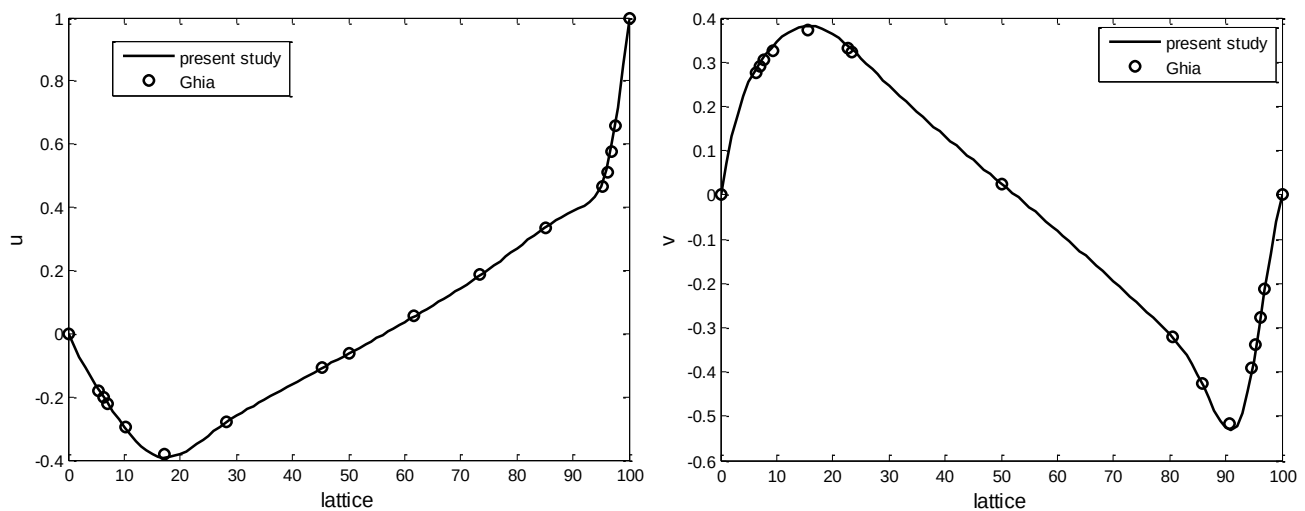


Figure 3. Validation of velocity in vertical lattice position: (a) x-direction (u) and (b) y-direction (v)

The visual stream functions and velocities quiver at $Re=500$ are shown on Figure 4. The stream functions show the same velocity as one line and the quivers show the flow directions. Both Figure 4(a) and 4(b) show that there are two vortices symmetrically. Logically there are two center of vortices that have zero velocity.

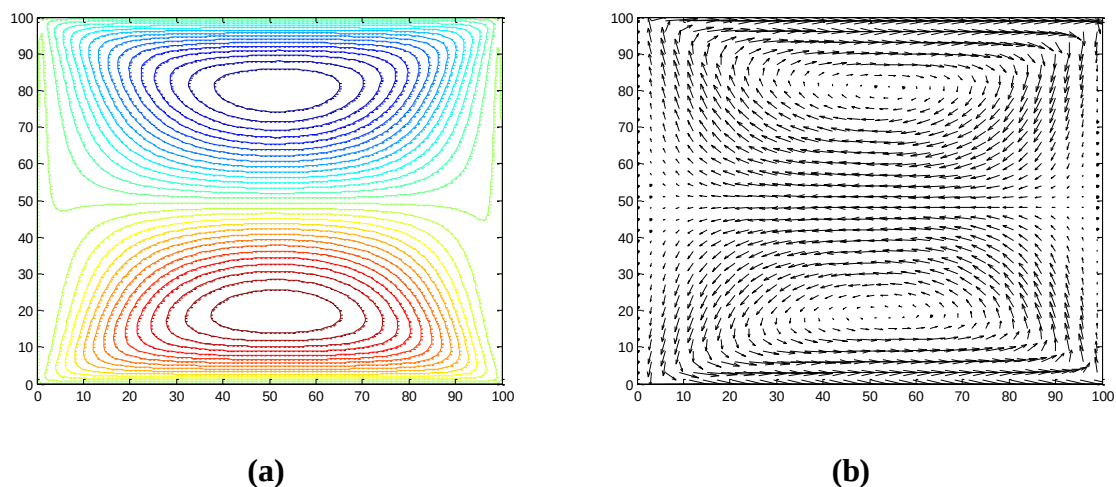


Figure 4. Visual at $Re=500$: (a) stream function and (b) quiver

The visual stream function and velocity quiver at $Re=1000$ are shown on Figure 5(a) and 5(b). Generally, the two-visual simulations at $Re=500$ and $Re=1000$ similar that there are two vortices. But if the plots are scrutinized, there are the differences in velocity and will be shown in the velocity plot in the end part of discussions.

The visual stream function and velocity quiver at $Re=2000$ are shown on Figure 6a(a) and 6(b). Two vortices still similar with the two previous Reynolds number. The detail differences also will be clearly shown on the next velocity plot vs. lattices. The interesting note that necessary to be mentioned is that at the right side of velocity stream in the center part of meeting point seems to be incipient of secondary vortex. So, if the investigations continue to run in the higher Reynolds number, at some extend will be found the secondary vortex.

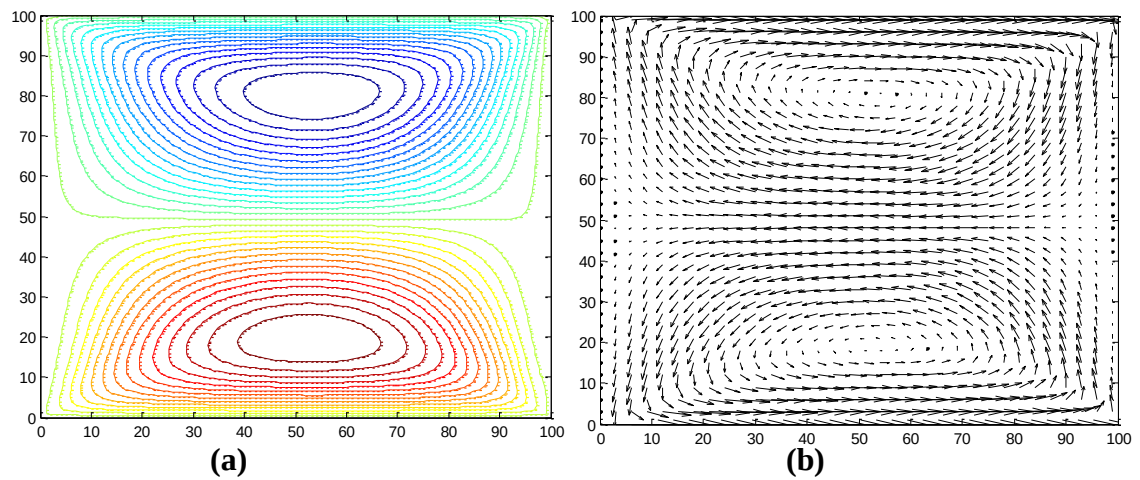


Figure 5. Visual at $Re=1000$: (a) stream function and (b) quiver

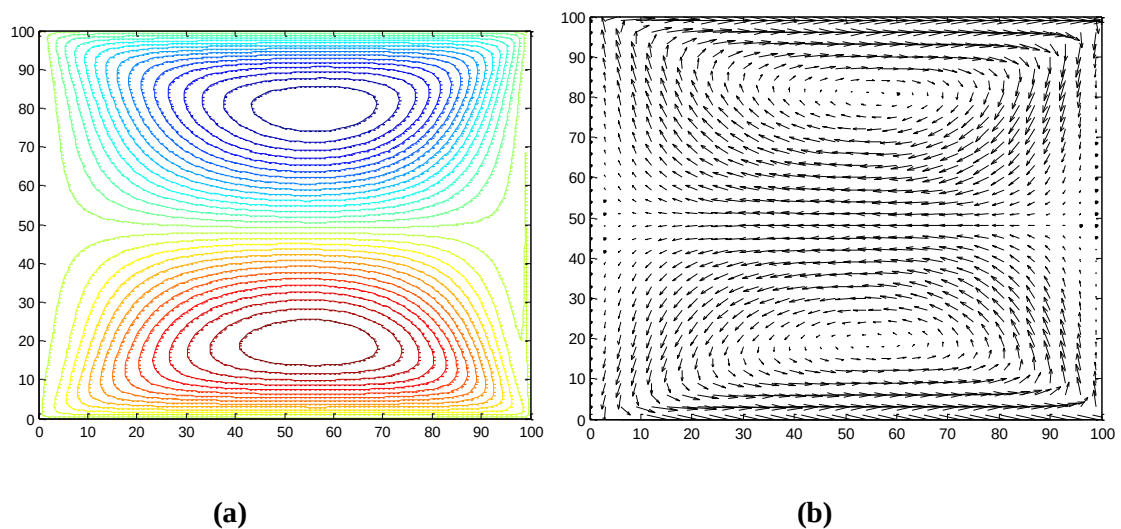


Figure 6. Visual at $Re=2000$: (a) stream function and (b) quiver

The next analysis is the comparison among the three Reynolds number upon the velocities of both directions: x-direction (u) and y direction (v). The locations where investigations run are at cross section of mid-horizontal of the cavity as shown on Figure 7.

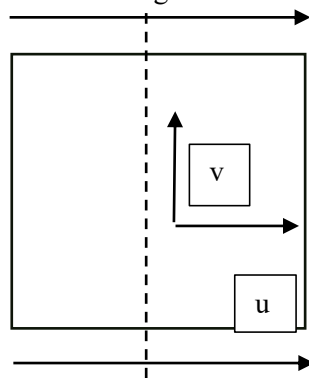


Figure 7. Positions of velocities of x and y-direction that were investigated

The plot results of x-direction velocity of all three Reynolds number is shown on Figure 8. As shown on the Figure 8, the velocity of x-direction plot in y-axis and lattice from bottom to top in x-axis.

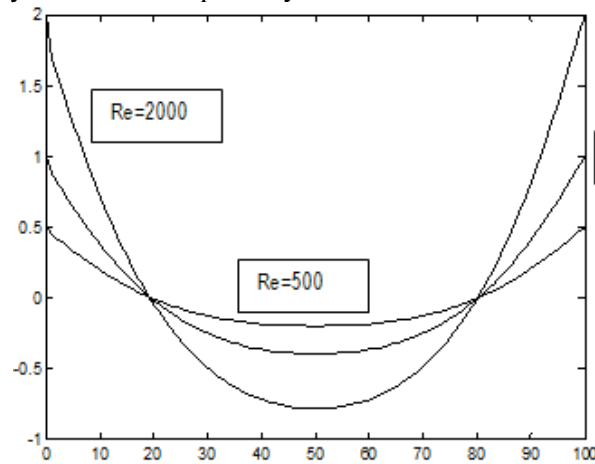


Figure 8. Plot of velocity of x-direction vs. lattice

Overall, the plots of all three Reynolds number are symmetrical with mirror at center of horizontal plane. The highest velocity in x-direction were located at lattice zero and lattice 100. These facts are true because at lattice zero and 100 locations of top and bottom lid that drive the flow. The zero velocity in x-direction case were located at around lattice 20 and 80 where center of vortices happen. The lowest position of all curves were located at the center or around lattice 50. That means the lowest velocity at x-direction is the highest velocity in the opposite direction with lid driven movement. This fact is confirmed where at the center of cavity the flow movement are to left direction and at this position is the location where two outer vortices met. From Figure 8 also can be found that the higher the Reynolds number, the steeper the curve. This fact can be explained that the higher Reynolds number in the same dimensions of geometry and the same fluid properties means the higher velocity of fluid flows.

The results of velocity in y-direction are plotted as shown on Figure 9. Overall can be seen from Figure 9 that the shape of the plots sinusoidal-like, with the zero velocity around at the middle. From bottom (lattice zero) until around the middle, the velocities in y-direction all positive or up-going. Then after passed zero point, the velocities in y-direction all negative or downward direction. From Figure 9 also can be found that the higher the Reynolds number, the steeper the curve.

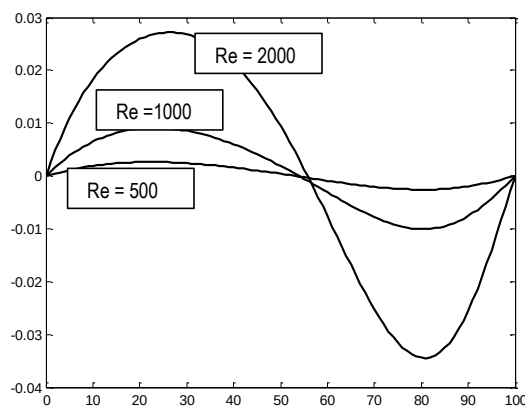


Figure 9. Plot of velocity of y-direction vs. lattice

4. Conclusion

Two main vortices happen in the case of top and bottom lid driven cavity that use Reynolds numbers are 500, 1000 and 2000. In the system of same dimension and physical fluid properties, the higher the Reynolds number, the steeper the curve velocity. The velocity in x-direction has quite symmetrical shape with the highest velocity in the nearest lid fluid and the lowest (or highest in opposite direction) in the middle of the cavity system. The velocity in the y-direction has sinusoidal-like curve shape with zero velocity around the middle of the cavity system.

Acknowledgement

This research is supported by Indonesia Endowment Fund for Education (Lembaga Pengelola Dana Pendidikan/LPDP) Gedung Danadyaksa Cikini Jl Cikini Raya No. 91A-D, Menteng, Jakarta Pusat 10330.

References

- [1] T. Lemée, G. Kasperski, G. Labrosse, and R. Narayanan, "Computers & Fluids Multiple stable solutions in the 2D symmetrical two-sided square lid-driven cavity," vol. **119**, pp. 204–212, 2015.
- [2] C. H. Marchi, R. Suero, and L. K. Araki, "The Lid-Driven Square Cavity Flow : Numerical Solution with a 1024 x 1024 Grid," vol. **XXXI**, no. 3, pp. 186–198, 2009.
- [3] T. A. Abdelmigid, K. M. Saqr, M. A. Kotb, and A. A. Aboelfarag, "Revisiting the lid-driven cavity flow problem : Review and new steady state benchmarking results using GPU accelerated code," *Alexandria Eng. J.*, vol. **56**, no. 1, pp. 123–135, 2016.
- [4] A. A. Mohammed, *Lattice Boltzmann Method: Fundamentals and Engineering Applications with Computer Codes*. 2012.
- [5] H. Huang, M. Sukop, and X. Lu, *Multiphase Lattice Boltzmann Methods Theory and Application*, vol. **1**. 2015.
- [6] T. Kruger, H. Kusumaatmaja, A. Kuzmin, O. Shardt, S. Goncalo, and E. M. Viggen, *The lattice boltzmann method, principles and practice*, no. 207. 2017.
- [7] U. Ghia, K. N. Ghia, and C. T. Shin, "High-Re Solutions for Incompressible Using the Navier-Stokes Equations Multigrid Method * Flow and a," *J. Comput. Phys.*, vol. **411**, pp. 387–411, 1982.