CHAPTER 3

SIMULATION OF NON-NEWTONIAN FLOW IN A LID DRIVEN CAVITY

The problem of fluid flow in lid driven cavity has been studied rigorously as a benchmark problem in CFD. However, most of these studies have been confined to either laminar fluid flow or Newtonian fluids in porous media. In this chapter, the problem of non-Newtonian fluid flow in a lid driven cavity is simulated using LBM. First, the governing equations in LBM are discussed along with its extension to non-Newtonian fluid flow in porous media. Later, these equations are solved to simulate non-Newtonian fluid flow in a lid driven cavity.
3.1. Lattice Boltzmann Method

We discussed earlier about evolution of LBM as an effective ‘mesoscopic’ numerical tool to study fluid flow in porous media. LBM can be looked upon as a Lagrangian finite-difference approximation to Boltzmann equations using a collision operator. The basic idea of this method is to identify the flow properties of a particle or a group of particles which act as the ‘representative’ for all those particles moving in a particular direction at a given time. The representative particles can move along pre-defined directions which are specified using lattice nodes. Particles shift from its position to a neighboring place (streaming), during which they may collide and change their directions (collision).

*Figure 3.1. A D2Q9 lattice model*

The direction along which the particles can move are specified based on the dimension (D) and the number of nodes (Q). For example, a D2Q9 lattice model is a two-dimensional nine-node lattice model as shown in Fig 3.1, which is the most frequently model used in simulating fluid flow in porous media. Streaming and collision follow some predefined rules based on the number of positions and directions.
Figure 3.2. (a) Amplified position of a D2Q9 lattice model (b) Distribution functions

Figure 3.3. Steaming of particles in a D2Q9 lattice model
One reason why LBM is simple to implement is that it reduces the number of particle spatial positions to the number of nodes of a lattice, which otherwise is large for a continuum. Other lattice models that are used in simulating fluid flow are D3Q15 and D3Q19, which are shown in Fig 3.4. A weight factor is assigned to each direction in a lattice model which resembles the probability of a particle moving along that direction. If the rest particles from D3Q15 and D3Q19 are removed, we get D3Q14 and D3Q18 models.

Figure 3.4. Examples of three-dimensional lattice models (a) D3Q15 model (b) D3Q19 model

Advantage of having these lattice models is in having freedom to choose the equilibrium distribution function and the structure of the simulation. In a D3Q15 model, the rest particles are included which increases accuracy of the algorithm. A higher lattice model increases the number of distribution functions contributing to the velocity components but also increases the number of iterations, computational time and the complexity of the algorithm. Sometimes lattice is represented by floating point numbers to reduce the memory usage.
while simulations, which otherwise may demand for extensive computational resources.

**3.1.1. Boltzmann equation to LBM-BGK approximation**

Let us say, $f_i(x,t)$ represents the proportion of particles at time $t$ positioned at $x$ moving with a velocity $e_i$. If there is no collision, the number of molecules before and after a time step will remain the same. Therefore,

$$f_i(x + e_i dt, t + dt) - f_i(x, t) = 0$$  \hspace{1cm} (3.1)

In case there is a collision, the relative difference between the distribution function is given by a collision operator, $\Omega$, and the evolution equation is given by (Mohamad [18])

$$f_i(x + e_i dt, t + dt) - f_i(x, t) = \Omega(f)$$  \hspace{1cm} (3.2)

The Boltzmann equation, as time spacing $dt \to 0$, is written

$$\frac{df}{dt} + c \cdot \nabla f = \Omega$$  \hspace{1cm} (3.3)

The collision operator, $\Omega$, is defined in terms of itself, making Boltzmann equation difficult to solve. Bhatnagar-Gross-Krook [25] approximated $\Omega$ as

$$\Omega = -\frac{1}{\tau}(f - f^{eq})$$  \hspace{1cm} (3.4)

Thus, the LBM-BGK equation is given by

$$f_i(x + e_i dt, t + dt) - f_i(x, t) = -\frac{1}{\tau}(f - f^{eq})$$  \hspace{1cm} (3.5)

where $f^{eq}$ is called the local equilibrium function, given by
\[ f_i^{eq}(\mathbf{x},t) = w_i \rho(\mathbf{x},t) \left[ 1 + \frac{1}{c_s^2} (\mathbf{e}_i \cdot \mathbf{u}) + \frac{1}{2 \varepsilon_s^2} (\mathbf{e}_i \cdot \mathbf{u})^2 - \frac{\mathbf{u} \cdot \mathbf{u}}{2 \varepsilon_s^2} \right] \] (3.6)

The expression on left side of Eq. 3.5 is the streaming part and that on the right side is the collision part. The relaxation parameter in the collision part tends to relax the velocity distribution function towards the local equilibrium. Although, they appear in a single equation, collision and streaming should be separated in order to consider boundary conditions. Since the streaming process is linear, it allows computation of non-linear macroscopic advection very easily, unlike in macro scale where this process is non-linear. The local equilibrium function along with the relaxation parameter decides the problem under investigation. The parameter, \( c_s = \frac{dx}{\sqrt{3}} dt = c / \sqrt{3} \) and \( w_i \) are weight coefficients given by

\[
w_i = \begin{cases} 4/9 & i = 0 \\ 1/9 & i = 1,2,3,4 \\ 1/36 & i = 5,6,7,8 \end{cases}
\] (3.7)

The particle speeds, \( \mathbf{e}_i \), are given by

\[
\mathbf{e}_i = \begin{cases} c(0,0) & i = 0 \\ c(\cos((i-1)\pi/2), \sin((i-1)\pi/2)) & i = 1,2,3,4 \\ \sqrt{2} c(\cos((i-5)\pi/2 + \pi/4), \sin((i-5)\pi/2 + \pi/4)) & i = 5,6,7,8 \end{cases}
\] (3.8)

Local particle density and local particle momentum are given by

\[
\rho(\mathbf{x},t) = \sum_{i=0}^{8} f_i(\mathbf{x},t)
\] (3.9)

\[
\rho u(\mathbf{x},t) = \sum_{i=0}^{8} \mathbf{e}_i f_i(\mathbf{x},t)
\] (3.10)

The pressure, \( p = \rho / 3 \) and the kinematic viscosity is given by
During each time step, the evolution of these particles takes place in three steps: collision, applying boundary conditions and streaming. In the first step, the distribution functions are calculated using Eq. 3.5. In the second step, the values of distribution functions for the particles moving towards the nodes at the boundary and outside the fluid are determined using boundary conditions. Finally in the third step, the particles stream to its neighboring nodes so that even the distribution functions are transferred to its adjacent nodes. There are several advantages of LBM. The most important among them is that it allows calculating pressure at local state, unlike the macroscopic approach which requires solving Poisson’s equation. Since LBM also helps to explain particle-fluid and fluid-structure interactions, it is very convenient to applications involving complex geometry and writing suitable programs for these purposes is very easy. The idea is to express macroscopic properties of fluid in terms of a collective behavior of particles, which simplifies the whole process of understanding the dynamics of fluid by constructing kinetic models to incorporate fluid properties at particle level so that averaged macroscopic properties, when expressed in terms of these mesoscopic properties, follow the macroscopic equations. Sometimes LBM is coupled with other an algorithm such as in (Guadarrama-Lara [44]) in which LBM is coupled with digital packing algorithm based on discrete element method to predict coupled interaction. Using Chapman-Enskog expansion, LBM can be related to the macroscopic parameters and Navier-Stokes equation can be recovered for small velocities and density variation (Peng et al. [45]), a derivation of which is discussed in Section 3.2.
Depending on the problem under investigation, a force term is added to the LBM equation and equilibrium distribution function specified, the procedure being much easier as compared to other methods. The procedure to solve any problem, whether it is in heat and mass transfer or diffusion, is the same; the only difference being the equilibrium distribution function and the relaxation parameter. Though LBM has its stability issues, application based conditions needs to be considered for it to be effective. For instance, in problems involving incompressible flow in porous media, where it is necessary that the Mach number (Ma) has to be less than the critical value of 1, the initial velocity is appropriately set for this condition to be satisfied. Stability of LBM also depends on the relaxation parameter, which along with the boundary conditions contributes highly in convergence of the algorithm. However, for problems involving non-Newtonian fluid flow in porous media the procedure of attaining convergence becomes very complex. This is because of the presence of additional parameters like power law index, consistency index and other constants depending on the model used. Also, the relaxation parameter in not fixed and varies locally to yield pore-level mechanism. The range of values of these parameters for convergence is sometimes very small, and thus, difficult to identify. This becomes a major difficulty when LBM is used to simulate non-Newtonian fluid flow and the success in identifying the values of these parameters determines the ability of achieving suitable results. Another aspect of this method is that the velocity components are calculated based on the distribution functions and its moments, as seen from Eq. 3.10. Thus, a number of distribution functions contribute to calculate these velocity components increasing the magnitude of error during computations. Since the algorithm runs over thousands of iteration, this error only gets magnified with the number of
iterations. Therefore, success in writing a good program depends on how best this error is reduced.

It is worthwhile to note that even though the objective is to find the velocity components, the dependent variables in the equation are the distribution functions. Thus, distribution functions are to be computed first in order to calculate the velocity components. This is one important factor among several others with the LBM. The lattice spacing (i.e. number nodes on the characteristic length) and time step also contribute to the stability and magnitude of error associated with the algorithm. Depending on the application, the number of lattice nodes has to be appropriately selected and the maximum number of grid points is decided based on the projected accuracy of the algorithm. For instance, in applications involving isothermal flows, the number of nodes is calculated to match Re and the aspect ratio using the formula

$$\text{Re} = \frac{UN}{\nu} \quad (3.12)$$

Therefore, if we choose $U = 0.01$, $\nu = 0.1$ and $\text{Re} = 10$, then the appropriate value for $N$ will be $\frac{10 \times 0.1}{0.01} = 100$.

However, the above estimation is based on the formulation of Re for Darcy flows as per Eq 3.12, which may be ineffective for non-Darcy regimes. One way of determining the optimum number of nodes for these flow regimes is to carry out the simulation for several sizes. The only difficulty in this process would be to distinguish the errors caused by grid size from errors caused by other factors like discretization of lattice, staggered invariants and compressibility. Nevertheless, some of these errors can be minimized by selecting appropriate flow parameters. For instance, errors caused due to number of nodes can be reduced by selecting smaller values of relaxation parameter (Ferreol and Rothmann [46]). In case of
geometries that have non-flat boundaries, discretization of lattice tends to approximate the boundary as smooth. Generally, this factor can be reduced by approximating such geometries by best possible porosity values. Compressibility errors occur due to variations in local density values based on variations in pressure which is found to increase for higher values of Reynolds number. In case of flow in porous media, averaging of density and other flow parameters reduces this error substantially (He and Luo [47]). In case of higher lattice models like D3Q15, where some of the nodes are not connected to its nearest neighbors, flow may result in irregular patterns depending on space and time. This error can be reduced by choosing appropriate initial conditions and averaging fluid velocity and density over a number of time steps.

3.1.2. Continuum limit of lattice Boltzmann method

In this section, we shall recover the Navier-Stokes equation from LBM governing equation. We start with the Taylor series expansion of Eq. 3.5, which will yield

$$\sum_{k=1}^{\infty} \frac{1}{k!} (\partial_t + e_i \partial_\alpha)^k f_i = - \frac{1}{\tau} (f - f^{eq})$$

(3.13)

The second order derivative approximation gives

$$\left( \partial_t + e_i \partial_\alpha \right) f_i^{eq} = \left( \frac{1}{2} \left( \partial_t^2 + 2 e_i \partial_t \partial_\alpha + e_i e_i e_\beta \partial_\alpha \partial_\beta \right) f_i^{eq} \right) + O\left( \partial^3 \right) = - \frac{1}{\tau} (f - f^{eq})$$

(3.14)

Taking a summation over $i$ in Eq. 3.13 and using Eq. 3.9-3.10

$$\partial_t \rho + \partial_\alpha (\rho u_\alpha) = \left( \frac{1}{2} \left[ \partial_t^2 + 2 \partial_t \partial_\alpha (\rho u_\alpha) + \partial_\alpha \partial_\beta \sum_i f_i^{eq} e_i e_i e_\beta \right] + \ldots \right) + O\left( \partial^3 \right) = 0$$

(3.15)

Multiplying Eq. 3.14 by $e_i e_\alpha$ and taking a summation over $i$, we get
\[
\partial_t (\rho u_\beta) + \partial_\alpha \sum_i f_i^{eq} e_{i\alpha} e_{i\beta} - \left( \tau - \frac{1}{2} \right) \partial_i^2 (\rho u_\beta) + \ldots
\]

\[
\ldots + 2 \partial_t \partial_\alpha \sum_i f_i^{eq} e_{i\alpha} e_{i\beta} + \partial_\alpha \partial_\gamma \sum_i f_i^{eq} e_{i\alpha} e_{i\beta} e_{i\gamma} \right) + O(\hat{\delta}^3) = 0
\] (3.16)

Eq. 3.16 is the continuity equation. Further, from Eq. 3.15 and Eq. 3.16 we get

\[
\partial_t \rho = -\partial_\alpha (\rho u_\alpha) + O(\hat{\delta}^2)
\] (3.17)

\[
\partial_t (\rho u_\beta) = -\partial_\alpha \sum_i f_i^{eq} e_{i\alpha} e_{i\beta} + O(\hat{\delta}^2)
\] (3.18)

Substituting these equations in Eq. 3.15, we get the continuity equation

\[
\partial_t \rho + \partial_\alpha (\rho u_\alpha) = 0 + O(\hat{\delta}^3)
\] (3.19)

Eq. 3.15 is equivalent to Navier-Stokes equation. Comparing Eq. 3.15 and Eq.3.17, we see that the term in the bracket in Eq. 3.15 can be neglected. The equilibrium distribution function \( f_i^{eq} \) is selected to make sure that the laws of conservation are satisfied. \( f_i^{eq} \) can be chosen as

\[
f_i^{eq} = A_k + B_k e_{i\alpha} u_\alpha + C_k u^2 + D_k (e_{i\alpha} u_\alpha)^2, k = 0,1,2.
\] (3.20)

If the first and third moment of the distribution function takes the form

\[
\sum_i f_i^{eq} = \rho(x,t), \quad \sum_i f_i^{eq} e_{i\alpha} = \rho(x,t) u_\alpha(x,t)
\] (3.21)

so that the density and momentum are conserved. The second moment of \( f_i^{eq} \) is taken as

\[
\sum_i f_i^{eq} e_{i\alpha} e_{i\beta} = p \delta_{\alpha\beta} + \rho u_\alpha u_\beta
\] (3.22)

Eq. 3.17 and Eq. 3.22 gives
\[
\frac{\partial}{\partial t} \sum_i f_i^{eq} e_i e_i = \frac{\partial}{\partial \gamma} \left( \rho u_\alpha u_\beta \right) + \frac{1}{3} \frac{\partial}{\partial \gamma} \left( \rho u_\gamma \delta_{\alpha\beta} + \rho u_\beta \delta_{\alpha\gamma} + \rho u_\alpha \delta_{\beta\gamma} \right)
\]

Using Eq. 3.18, the term inside the bracket in Eq. 3.15 can be written as

\[
\frac{\partial}{\partial t} \sum_i f_i^{eq} e_i e_i e_i = \frac{1}{3} \frac{\partial}{\partial \gamma} \left( \rho u_\gamma \delta_{\alpha\beta} + \rho u_\beta \delta_{\alpha\gamma} + \rho u_\alpha \delta_{\beta\gamma} \right)
\]

Substituting Eq. 3.23 and Eq. 3.24 in Eq. 3.15, we get

\[
\frac{\partial}{\partial t} (\rho u_\beta) + \frac{\partial}{\partial \alpha} \left( \rho u_\alpha u_\beta \right) = \frac{\partial}{\partial \beta} p + \frac{3}{\tau} \left[ \frac{\partial}{\partial \alpha} \left( u_\alpha u_\beta + \rho u_\alpha \right) + \frac{1}{3} \delta_{\alpha\beta} \delta_{\gamma\gamma} \right]
\]

which is the Navier-Stokes equation that also includes the kinematic viscosity

\[
\nu = -\frac{1}{\tau}, \text{ same as in Eq. 3.11.}
\]

### 3.1.3. Lattice Boltzmann method for fluid flow in porous media

In general, fluid flow in porous media is governed by equations discussed in Chapter 2 (Eq. 2.1 and Eq. 2.4). However, the effect of porous material to fluid flow is included by adding total body force due to porous media (Ergun [4], Peng [45], He and Luo [47], Seta et al. [48]), which along with the inertial force also includes the viscous force and external force giving

\[
F = -\frac{\nu}{K} u - \frac{\varepsilon F_s}{\sqrt{K}} \|u\| + \varepsilon G
\]

A suitable choice for \( F \) in discrete form is discussed in [Guo and Zhao [20], Mehrizi et al. [49], Peng et al. [50]) given by

\[
F_i = w_i \rho \left( 1 - \frac{1}{2\tau} \left[ \frac{e_i F}{c_s^2} + \frac{u F \cdot (e_i e_i - c_s^2 I)}{\varepsilon c_s^2} \right] \right)
\]

The total force in Eq. 3.27 defines fluid velocity as (Seta et al [51])
which is non-linear in nature. Guo and Zhao [20] simplified the process of finding $u$ by introducing a temporal velocity $u_t$.

\[ u_t = \sum_i e_i f_i + \frac{dt}{2} \rho \varepsilon G \]  

(3.29)

Therefore, $u$ is given by

\[ u = \frac{u_t}{c_0 + \sqrt{c_0^2 + c_1 |u_t|}} \]  

(3.30)

where the two constants $c_0$ and $c_1$ are defined as

\[ c_0 = \frac{1}{2} \left( 1 + \varepsilon \frac{dt}{2} \frac{\nu}{K} \right), \quad c_1 = \varepsilon \frac{dt}{2} \frac{F_{\varepsilon}}{\sqrt{K}} \]  

(3.31)

The LBM-BGK equation is accordingly modified to include the total body force and written as

\[ f_i(x + e_i dt, t + dt) - f_i(x, t) = \frac{f_i^{eq}(x, t) - f_i(x, t)}{\tau} + dt f_i \]  

(3.32)

For simulation of non-isothermal flows, energy equation is added to the system of governing equations given by

The energy equation to model heat transfer is written as (Peng et al. [45])

\[ g_i(x + e_i dt, t + dt) - g_i(x, t) = \frac{g_i^{eq}(x, t) - g_i(x, t)}{\tau_g} \]  

(3.33)

The power-law model in Eq. 2.9 can be used to identify local viscosity of a fluid based on the variation in local strain rate, whereas Eq. 3.11 expresses local
viscosity in terms of $\tau$, which is either constant for a Newtonian fluid or varies locally based on local strain rate for a non-Newtonian fluid. Thus, a systematic procedure to simulate fluid flow using LBM can be detailed as follows:

1. Set initial values of the velocity distribution function $f_i$ and the equilibrium distribution function $f_i^{eq}$.
2. Calculate $\rho$ using Eq. 3.9 and the velocities using Eq. 3.10.
3. Calculate $f_i^{eq}$ using Eq. 3.6 and determine $\gamma$.
4. Calculate $\nu$ using Eq. 2.9 and then $\tau$ using Eq. 3.11.
5. Calculate total body force using Eq. 3.15 to calculate the velocity distribution function $f_i$ using Eq. 3.20.
6. Carry out streaming step to move the particles to their neighboring nodes followed by the collision step based on the boundary conditions.
7. Repeat the procedure from Step 2 until the convergence criteria is satisfied.

3.1.4. Multiple-Relaxation-Time lattice Boltzmann method

LBM-BGK, also referred to as single-relaxation time (SRT-LBM) model, is the most popular model among the lattice Boltzmann methods for its simplicity in implementations. Nevertheless, it has shown instability in simulating fluid flow at low viscosities as discussed in (Peng et al. [50], Seta et al. [51]). Thus, LBM-BGK was extended to a multiple-relaxation-time (MRT-LBM) by d’Humeires et al. [52] to overcome this instability. As a result, number of studies has used MRT-LBM to simulate fluid flow because of its stability and accuracy (Lallemand and Luo [53], Chai et al. [54], Fallah et al. [55], Yang et al. [56], Sterling and Chen [57]). The basic idea of the MRT-LBM is to generalize the collision operator $\Omega$ to a collision
matrix and by performing the collision process in momentum space. Eq. 3.2, therefore, is transformed into

$$f_i(x + e_i dt, t + dt) - f_i(x, t) = -M^{-1} R [m(x, t) - m^{eq}(x, t)]$$  \hspace{1cm} (3.34)

where $m = (m_0, m_1, m_2, ..., m_n)^T$ and $m^{eq}$ moment vectors, $R$ is a diagonal matrix.

For a D2Q9 model, the matrix $m$ is given by

$$m = \begin{bmatrix}
1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1
-4 & -1 & -1 & -1 & -1 & 2 & 2 & 2 & 2
4 & -2 & -2 & -2 & -2 & 1 & 1 & 1 & 1
0 & 1 & 0 & -1 & 0 & 1 & -1 & -1 & 1
0 & -2 & 0 & 2 & 0 & 1 & -1 & -1 & 1
0 & 0 & 1 & 0 & -1 & 1 & 1 & -1 & -1
0 & 0 & -2 & 0 & 2 & 1 & 1 & -1 & -1
0 & 1 & -1 & 1 & -1 & 0 & 0 & 0 & 0
0 & 0 & 0 & 0 & 0 & 1 & -1 & 1 & -1
\end{bmatrix}$$

The equilibrium moment $m^{eq}$ is given by

$$m^{eq} = (m_0^{eq}, m_1^{eq}, m_2^{eq}, m_3^{eq}, m_4^{eq}, m_5^{eq}, m_6^{eq}, m_7^{eq}, m_8^{eq})$$

$$= (\rho, -2\rho, 3(j_x^2 + j_y^2), \rho - 3(j_x^2 + j_y^2), j_x, -j_x, j_y, -j_y, j_x^2 - j_y^2, j_x j_y)$$  \hspace{1cm} (3.35)

where $j_x = \sum_i f_i^{eq} e_i(x)$ and $j_y = \sum_i f_i^{eq} e_i(y)$. The collision diagonal matrix $R$ is given by

$$R = \text{diag}\left(1.0, 1.4, 1.4, 1.0, 1.0, 1.2, 1.0, 1.2, \frac{2}{1+6u_y}, \frac{2}{1+6u_y} \right).$$  \hspace{1cm} (3.36)

In a SRT-LBM, $R$ is the collision operator diagonal matrix with non-zero elements as $\frac{1}{\tau}$. The procedure for MRT-LBM, otherwise, remains the same as that for SRT-LBM.
3.2. Simulation of non-Newtonian fluid flow in a lid driven cavity

The advantage of using LBM to simulate non-Newtonian fluid flow shall be two folds: first, that the numerical procedure of LBM will be validated for Newtonian fluids with respect to results in literature; and second, the mechanism of non-Newtonian fluids can be explored which has high importance in various natural and engineering processes such as in packed beds, petroleum engineering and purification processes.

In general, the hydrodynamics of non-Newtonian fluids is much complex compared to that of their Newtonian counterpart because of the complex rheological properties. An important factor in understanding the mechanism of non-Newtonian fluids is to identify a local profile of non-Newtonian properties corresponding to the shear rate. Power law model is generally used to represent a class of non-Newtonian fluids which are inelastic and exhibit time independent shear stress. Though analytical solutions for the flow of power law fluids through simple geometry is available, computational approach becomes unavoidable in most of the situations particularly if the flow field is not one dimensional. Over the past several years, various computational methods have been applied to simulate the power law fluid flows in different geometries. Among different geometries, the lid driven cavity flow is considered to be one of the benchmark fluid problems in computational fluid dynamics. A good research on Newtonian and non-porous lid driven flow is available in Guo and Zhao [20], Patil et al. [58], Bhaumik and Lakshmisha [59], Menati et al. [60], Erturk [61], Mendu and Das [62] and Perumal et al. [63].

3.2.1. Problem Description

LBM is used to simulate lid driven cavity flow of non-Newtonian fluids which is laminar as well as under transition for a wide range of shear-thinning and shear-
thickening fluids. The power law model is used to represent the class of non-Newtonian fluids (shear-thinning and shear-thickening). The influence of power law index \( n \) and Reynolds number \( (Re) \) on the variation of velocity and center of vortex location of fluid with the help of velocity profiles and streamline plots is studied.

![Figure 3.5. Geometry of the lid driven cavity](image)

A two-dimensional lid driven square cavity containing porous media with the top wall moving from left to right with a uniform velocity \( U = u_0 = 0.1 \) is considered, as shown in Fig 3.5. The left, right and bottom walls are kept stationary i.e. velocities at all other nodes are set to zero. Half-way bounce-back conditions are applied on the stationary walls. At the lattice nodes on the moving walls, boundary conditions are assumed as specified by Zou and He [64]. Initially, the equilibrium distribution function that corresponds to the flow-variables is assumed as the unknown distribution function for all nodes at \( t = 0 \). The governing equations specified in Eq. 3.5-3.11 are solved using an iterative algorithm by a MATLAB code developed for a \( 129 \times 129 \) square cavity lattice grid. The velocity of the moving lid was set to 0.1 and the velocity at all other nodes was set to zero. A uniform density of \( \rho = 1.0 \) with fixed porosity and permeability is initially assumed for the entire flow field.
The distribution function was initialized with suitable values (here it is assumed that fluid is initially stationary). A range of 0.8 to 1.6 was taken for the power law index, so that, both the shear thinning and shear-thickening fluid are considered. The investigation is performed for four values of Re (100, 400, 1000 and 3200).

3.2.2. Results and Discussion

Fig 3.6 presents velocity profiles at the geometric center of the cavity for various values of $n$ at $Re = 100$ along with the comparison of the results with Ghia et al. [65] for Newtonian fluids by taking $\varepsilon = 0.999$ corresponding to non-porous flow in case of Newtonian fluids. The results show a very good agreement with the results in literature, validating the use of LBM as computational technique to simulate non-Newtonian fluid flow. The basic trend of the $u$-velocity profiles remain the same as Re is increased from 100 to 3600 as seen in Fig 3.6-3.9, except that the rheological behavior of the fluids affects the minimum and the maximum values of the $u$-velocity profiles. Lower values of $n$ (shear-thinning fluid and Newtonian) display lower values of $v$-velocity compared to that of shear-thickening fluids.

For lower Re, the $u$-velocity profiles are seen to be curved even at the center part of the cavity but become flatter for higher values of Re, which means the velocity at the vertical center decreases smoothly. In general, velocities first take a negative trend starting from zero and then become zero at the center moving towards positive values at upper part converging towards one. However, a slight variation is observed in $u$-velocity profile for $Re = 3200$ as $u$-velocity at the bottom of the cavity first takes positive values after which it moves towards a negative trend. Similar behavior is observed for $v$-velocity profiles with the peak values shifting towards the extreme ends of the vertical side as Re increases from 100 to 3200.
Figure 3.6. Velocity profiles at the geometric center of the cavity for different values of $n$ at $Re=100$ (a) $u$-velocity profiles (b) $v$-velocity profiles.
Figure 3.7. Velocity profiles at center of cavity for various values of $n$ at $Re=400$ (a) $u$-velocity profiles (b) $v$-velocity profiles.
Figure 3.8. Velocity profiles at the center of the cavity for various values of $n$ at $Re=1000$ (a) u-velocity profiles (b) v-velocity profiles.
Figure 3.9. Velocity profiles at center of the cavity for values of n at Re=3200 (a) u-velocity profiles (b) v-velocity profiles.
Figure 3.10. Streamline plots for various values of $n$ at $Re=100$ (a) $n=0.8$ (b) $n=1.0$ (c) $n=1.2$ (d) $n=1.4$ (e) $n=1.6$. 
Figure 3.11. Streamline plots for various values of $n$ at $Re=400$ (a) $n=0.8$ (b) $n=1.0$ (c) $n=1.2$ (d) $n=1.4$ (e) $n=1.6$. 
Figure 3.12. Streamline plots for various values of n at Re=1000 (a) n=0.8 (b) n=1.0 (c) n=1.2 (d) n=1.4 (e) n=1.6.
Figure 3.13. Streamline plots for various values of $n$ at $Re=3200$ (a) $n=0.8$ (b) $n=1.0$ (c) $n=1.2$ (d) $n=1.4$ (e) $n=1.6$. 
The magnitude of the $u$-velocity and $v$-velocity profiles increases, which is also due to the impact of moving lid. Along with the moving lid as $Re$ increases, it results in higher velocity at the upper half of the velocity compensating by decreasing the velocity at bottom of the cavity towards negative minimum. In the transition phase, the negative velocity at the bottom induces an opposite fluid movement resulting in a small positive velocity profile as seen in Fig 3.9a. These movements are responsible for secondary flow. Fig. 3.10-3.13 presents the streamline plots for different values of power law index at $Re = 100$. One primary vortex is observed which moves towards the center of the cavity as $n$ increases from 0.8 to 1.6. This is because the viscosity of the fluid increases with increasing $n$. Secondary vortices are observed at the top and bottom corners of the cavity. These vortices decrease marginally and almost vanish with an increase in $n$, thus indicating that the primary circulation occupies almost whole of the cavity. The secondary circulation increases as compared to lower values of $Re$. This is in theoretical agreement of the properties of fluids, that the secondary circulation and formation of eddies are closely related to increase in $Re$. The magnitude of the $u$-velocity and $v$-velocity profiles increases, which is also due to the impact of moving lid. The results also establish the validity of LBM to simulate non-Newtonian fluid flow and provide an effective numerical tool that helps in understanding micro-level particle behavior of fluid.