ABSTRACT

Lattice Boltzmann Method (LBM) is used to simulate the lid driven cavity flow to explore the mechanism of non-Newtonian fluid flow. The power law model is used to represent the class of non-Newtonian fluids (shear-thinning and shear-thickening fluids) by considering a range of 0.8 to 1.6. Investigation is carried out to study the influence of power law index and Reynolds number on the variation of velocity profiles and streamlines plots. Velocity profiles and the streamline patterns for various values of power law index at Reynolds numbers ranging 100 to 3200 are presented. Half way bounce back boundary conditions are employed in the numerical method. The LBM code is validated against the results taken from the published sources for flow in lid driven cavity and the results show fine agreement with established theory and the rheological behavior of the fluids.

Keywords: Lid Driven Cavity, Non-Newtonian Fluids, Power Law, Lattice Boltzmann Method.

1. INTRODUCTION

Non-Newtonian fluid flow is an important subject in various natural and engineering processes which include applications in packed beds, petroleum engineering and purification processes. A wide range of research is available for Newtonian and non-Newtonian fluid flow in these areas. 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.

With recent advances in mathematical modeling and computer technology, lattice Boltzmann method (LBM) has been developed as an alternative approach to common numerical methods which are based on discretization of macroscopic continuum equations. LBM is effective for investigating the local non-Newtonian properties since important information to non-Newtonian fluids can be locally estimated. The fundamental idea of this method is to construct simplified kinetic models that include the essential physics of mesoscopic processes so that the macroscopic averaged properties obey the desired macroscopic properties [2].

In recent years, the problem of lid driven cavity flow has been widely used to understand the behavior of non-Newtonian fluid flow using power law model. Patil et al [5] applied the LBM for simulation of lid-driven flow in a two-dimensional, rectangular, deep cavity. They studied the location and strength of the primary vortex, the corner-eddy dynamics and showed the existence of corner eddies at the bottom, which come together to form a second primary-eddy as the cavity aspect-ratio is increased above a critical value. Bhuiyan et al [6] investigated lid-driven swirling flow in a confined cylindrical cavity using LBM by studying steady, 3-dimensional flow with respect to height-to-radius ratios and Reynolds numbers using the multiple-relaxation-time method. Nemati et al [7] applied LBM to investigate the mixed convection flows utilizing nanofluids in a lid-driven cavity. They investigated a water-based nanofluid containing Cu, CuO or Al₂O₃ nanoparticles and the effects of Reynolds number and solid volume fraction for different nanofluids on hydrodynamic and thermal characteristics. Mendu et al. [9] used LBM to simulate non-Newtonian power law fluid flows in a double sided lid driven cavity. They investigated two different cases-parallel wall motion and anti-parallel wall motion of two sided lid driven cavity and studied 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. In another paper, Mendu et al. [10] applied LBM to simulate two dimensional fluid flows in a square cavity driven by a periodically oscillating lid. Yang et al. [25] investigated the flow pattern in a two-dimensional lid-driven semi-circular cavity based on multiple relaxation time lattice Boltzmann method (MRT LBM) for Reynolds number ranging from 5000 to 50000. They showed that, as Reynolds number increases, the flow in the cavity undergoes a complex transition. Erturk [8] discussed, in detail, the 2-D driven cavity flow problem by investigating the incompressible flow in a 2-D driven cavity in terms of physical, mathematical and numerical aspects, together with a survey on experimental and numerical studies. The paper also presented very fine grid steady solutions of the driven cavity flow at very high Reynolds numbers.

The application of LBM to non-Newtonian fluid flow has been aggressively intensified in last few decades. Though, the problem of fluid flow in lid driven cavity has been studied rigorously, most of these studies have been confined to either laminar fluid flow or Newtonian fluids. The present paper uses LBM to simulate the lid driven cavity flow to explore the mechanism of non-Newtonian fluid flow 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.
Fig.1: Geometry of the lid driven cavity.

2. BACKGROUND AND PROBLEM FORMULATION

2.1. Governing equations

In continuum domain, fluid flow is governed by Navier-Stokes (NS) equations along with the continuity equation. For incompressible, two-dimensional flow, the conservative form of the NS equations and the continuity equation can be written in Cartesian system as [1].

\[
\begin{align*}
\frac{\partial \rho u}{\partial t} + \frac{\partial \rho u u}{\partial x} + \frac{\partial \rho u v}{\partial y} &= -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left( \mu \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu \frac{\partial u}{\partial y} \right) \\
\frac{\partial \rho v}{\partial t} + \frac{\partial \rho u v}{\partial x} + \frac{\partial \rho v v}{\partial y} &= -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left( \mu \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu \frac{\partial v}{\partial y} \right) \\
\frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} &= 0
\end{align*}
\]  

The two-dimensional lid driven square cavity with the top wall moving from left to right with a uniform velocity \( U = u_0 = 0.1 \) is considered, as shown in Fig 1. The left, right and bottom walls are kept stationary i.e. velocities at all other nodes are set to zero. We consider fluid to be non-Newtonian represented by power law model.

2.2. Physical boundary conditions

Top moving lid : \( u(x, H) = u_0 \) and \( v(x, H) = 0 \)  
Bottom stationary side : \( u(x,0) = 0 \) and \( v(x,0) = 0 \)  
Left stationary side: \( u(0, y) = 0 \) and \( v(0, y) = 0 \)  
Right stationary side: \( u(L, y) = 0 \) and \( v(L, y) = 0 \)
3. NUMERICAL METHOD AND FORMULATION

3.1. Lattice Boltzmann Method

In the present study, we cover incompressible fluid flows and a nine-velocity model on a two-dimensional lattice (D2Q9). The lattice Boltzmann method can be used to model hydrodynamic or mass transport phenomena by describing the particle distribution function \( f_i(x,t) \) giving the probability that a fluid particle with velocity \( e_i \) enters the lattice site \( x \) at a time \( t \) \([1]\). The subscript \( i \) represents the number of lattice links and \( i = 0 \) corresponds to the particle at rest residing at the center. The evolution of the particle distribution function on the lattice resulting from the collision processes and the particle propagation is governed by the discrete Boltzmann equation \([11, 12, 13, 14]\).

\[
f_i(x + e_idt, t + dt) - f_i(x, t) = \Omega_i(x, t) \quad i = 0, 1, \ldots, 8 \quad (5)
\]

where \( dt \) is the time step and \( \Omega_i \) is the collision operator which accounts for the change in the distribution function due to the collisions. The Bhatnagar-Gross-Krook (BGK) model \([19]\) is used for the collision operator

\[
\Omega_i(x, t) = -\frac{1}{\tau} \left[ f_i(x, t) - f_i^\text{eq}(x, t) \right] \quad i = 0, 1, \ldots, 8 \quad (6)
\]

where \( \tau \) is the relaxation time and is related to the kinematic viscosity \( \nu \) by

\[
\nu = c_s^2 dt \left( \tau - \frac{1}{2} \right) \quad (7)
\]

Here \( c_s \) is the sound speed expressed by \( c_s = dx / \sqrt{3} \) \( dt = c / \sqrt{3} \) (\( c \) is the particle speed and \( dx \) is the lattice spacing). \( f_i^\text{eq}(x, t) \), in equation (6), is the corresponding equilibrium distribution function for D2Q9 given by

\[
f_i^\text{eq}(x, t) = w_i \rho(x, t) \left[ 1 + \frac{1}{2c_s^2} (e_i u(x, t))^2 + \frac{1}{2c_s^2} (e_i u(x, t))^2 - \frac{1}{2c_s^2} (u(x, t),u(x, t)) \right] \quad (8a)
\]

where \( u(x, t) \) is the velocity and \( w_i \) is the weight coefficient with values

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

Local particle density \( \rho(x, t) \) and local particle momentum \( \rho \nu \) are given by

\[
\rho(x, t) = \sum_{i=0}^{8} f_i(x, t) \text{ and } \rho \nu(x, t) = \sum_{i=0}^{8} e_i f_i(x, t) \quad . \quad (9)
\]
3.2. Power law model

For a power law fluid, the apparent viscosity $\mu$ is found to vary with strain rate $\varepsilon (s^{-1})$ by [15]

$$\mu = \mu_0 \varepsilon^{-n}$$

(10)

where $n$ is the shear-thinning index and $\mu_0$ is a consistency constant. For $n<1$, the fluid is shear-thinning. Strain rate is related to the symmetric strain rate tensor,

$$\dot{e} = 2\sqrt{e_{\alpha\beta} : e_{\alpha\beta}}$$

(11a)

where $e_{\alpha\beta}$ is the rate of deformation tensor which can be locally calculated by [11]

$$e_{\alpha\beta} = -\frac{1}{2C_\tau} \sum_{\tau=0}^{8} f_i^{(1)} e_{\tau\alpha} e_{\beta}$$

(11b)

where $f_i^{(1)}(x,t) = f_i(x,t) - f_i^{eq}(x,t)$ is the non-equilibrium part of the distribution function.

The Reynolds number for power law fluid is given by

$$Re = \frac{\mu U^2}{\rho L^n},$$

where $U$ is the velocity of the moving lid and $L$ is the length of the cavity [9].

3.3. LBM boundary conditions

Boundary conditions play a crucial role in LBM simulations. In this paper, half-way bounce-back conditions [17] are applied on the stationary wall. The particle distribution function at the wall lattice node is assigned to be the particle distribution function of its opposite direction. At the lattice nodes on the moving walls, boundary conditions are assumed as specified by Zou et al. [4]. Initially, the equilibrium distribution function that corresponds to the flow-variables is assumed as the unknown distribution function for all nodes at $t = 0$.

3.4. Numerical Implementation

A MATLAB code was developed for a $129 \times 129$ square cavity lattice grid. The velocity of the moving lid was set to 0.1 $lu/ls$ and the velocity at all other nodes was set to zero. A uniform density of $\rho = 1.0 m/u/lu^3$ is initially assumed for the entire flow field. The distribution function was initialized with suitable values (here we assume that the fluid is initially stationary). The numerical implementation of the LBM at each time step consists of collision, streaming, application of boundary conditions, calculation of distribution functions and calculation of macroscopic flow variables. Half way bounce back conditions were employed in simulation.

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.

4. RESULTS AND DISCUSSIONS

Investigation is performed to study the impact of Reynolds number and power index on velocity profiles, vortex formation and the streamline patterns. The LBM was first validated by comparing the results in the literature for power index $n = 1$, which corresponds to Newtonian fluids.
The circulation pattern and vortex formation is highly influenced by Reynolds number. The investigation is performed for four values of Re (100, 400, 1000, 3200). Fig.2 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 published in Ghia et al [3] for Newtonian fluids. The u-velocity profiles along y-axis are presented in Fig.2a, which show almost a parabolic behavior. It is observed that u-velocity starts increasing from zero at the bottom, continuously decreasing to the minimum negative value and then increases to become zero at the center. The u-velocity then increases to attain the maximum positive value at the top of the cavity. The minimum u-velocity value is observed to decrease with an increase in $n$, whereas the maximum positive u-velocity increases with $n$. Though the basic trend of the u-velocity profiles remain the same, the rheological behavior of the fluids affects the minimum and the maximum values of the u-velocity profiles. Fig.2b presents the v-velocity profile along x-axis through the geometric center of the cavity corresponding to the conditions considered in Fig.2a. It is observed that the v-velocity starts from zero at the left side wall, attains the maximum positive value, and then decreases to zero at the center of the cavity. The v-velocity continuously decreases from the center and reaches the maximum negative value before it again tends to zero at the right wall of the cavity. Lower values of $n$ (shear-thinning fluid and Newtonian) display lower values of v-velocity compared to that of shear-thickening fluids. Fig.4 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. Primary circulation is concentrated towards the right of the cavity with streamlines parallel to the right vertical side, indicating weak circulation in the region.

Fig.3 presents the velocity profiles at geometric center of the cavity for different values of $n$ at Re = 400. The velocity at the center of the cavity becomes almost linear with maximum and minimum values attained at the top and bottom regions of the cavity, respectively. The streamline plots in Fig. 5 for Re = 400 show development of secondary vortices at the corners which keep increasing in size with $n$, indicating the growth of secondary circulation in the regions. The primary vortex is observed to shift towards the center as compared to the case for Re = 100. For Re=1000, the trend of the velocity profiles remains almost the same, but a significant variation is observed in magnitude as indicated by Fig.7. This is due to the influence of moving lid on the fluid flow. 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 closely related to increase in Re. Fig.6 shows that the strength of the secondary circulation increases as $n$ increases from 0.8 to 1.6 It can be seen from Fig.3-7 that, for lower values of Re, the streamlines are concentrated parallel to the right wall due to weak circulation in the region. At higher values of Re, the primary circulation expands to cover whole of cavity indicating stronger circulation in the regions.

The primary vortex shifts towards the center of the cavity for higher Re. The u-velocity and v-velocity profiles for Re = 3200 are presented in Fig.8. The profiles show a linear behavior in center of the cavity, as seen from the figures. The u-velocity profiles for higher values of $n$ show positive u-velocities at the bottom of the cavity, as a result of strong secondary circulation in opposite direction. This can be established from the streamline plots in Fig.9, which show formation of strong secondary circulation as a result of transitional state of fluid. The velocity profiles show a linear behavior in the centre of the cavity and this region (of linear behavior) increases as Re increases from 100 to 3200. The magnitude of the u-velocity and v-velocity profiles increases, which is also due to the impact of moving lid. Secondary circulation is more dominant compared to other values of Re, indicating the influence of Re.
Fig. 2: 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

Fig. 3: Velocity profiles at center of cavity for various values of $n$ at $Re=400$. (a) $u$-velocity profiles (b) $v$-velocity profiles
Fig. 4: 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) $n=1.6$
Fig.5: 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
Fig.6: 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) $n=1.6$
Fig. 7: Velocity profiles at the center of the cavity for various values of $n$ at Re=1000.
(a) u-velocity profiles (b) v-velocity profiles

Fig. 8: Velocity profiles at center of the cavity for values of $n$ at Re=3200
(a) u-velocity profiles (b) v-velocity profiles
Fig. 9: Streamline plots for various values of $n$ at $Re=3200$. (a) $n=0.8$ (b) $n=1$ (c) $n=1.2$ (d) $n=1.4$ (e) $n=1.6$
5. CONCLUSION

The paper presents numeric simulation of non-Newtonian (shear-thinning and shear-thickening) fluid flow in a lid-driven square cavity. The influence of power law index and Reynolds number on velocity profiles and the streamline plots is presented. The simulation is performed for four values of Re (100, 400, 1000, 3200) and a range of 0.8 to 1.6 for power index. The Re has high impact on the velocity profiles and the formation of the primary vortex. The u-velocity profiles which show a parabolic behavior for smaller values of Re, become almost linear at the center of the cavity for higher values of Re. The strength of the secondary vortices is also influenced by Re, the strength of which increases with Re in agreement to the rheological behavior of the fluid. The position of the primary vortex is highly influenced by the power law index and the Reynolds number, which shifts towards the center of the cavity with increasing values of Re. The study also demonstrates LBM to be an effective and alternative numerical method to simulate non-Newtonian fluid flow.

REFERENCES


