Prediction of Natural Convection in a Square Cavity with Partially Heated from Below and Symmetrical Cooling from Sides by the Finite Difference Lattice Boltzmann Method

Nor Azwadi Che Sidik  
Thermo-fluid Department, Universiti Teknologi Malaysia  
UTM Skudai, Johor, Malaysia  
E-mail: azwadi@fkm.utm.my  
Tel: +60-7-5534718; Fax: +60-7-5566169

Abstract

In this paper, the lattice Boltzmann method, a numerical tool based on the particle distribution function is used to simulate thermal fluid flow problem. A well-known finite difference technique with third order accuracy in space is combined with the double population thermal lattice Boltzmann model to solve two-dimensional natural convection in a square cavity. The basic idea is to solve the velocity and the temperature field using two different distribution functions. The simulation of localized heated bottom and symmetrical cooling from the sides were carried out in order to validate this approach. The combination of finite difference with double population thermal lattice Boltzmann model is found to be an efficient and stable numerical approach for low and moderate Rayleigh number calculations.

Keywords: Lattice Boltzmann, Double Population, Finite Difference, Natural Convection, Localised Heating

1. Introduction

For more than a decade, lattice Boltzmann method (LBM) has been demonstrated to be a very effective numerical tool for a broad variety of complex fluid flow phenomena that are problematic for conventional method (Bernsdorf, Brenner and Durst, 2000; Nor Azwadi and Tanahashi, 2006). Compared with traditional computational fluid dynamics, LBM algorithms are much easier to be implemented especially in complex geometries and multicomponent flows (Michael, Orlandini, Orborn and Yeomans, 1996).

Historically, LBM was derived from lattice gas (LG) automata (Frish, Hasslacher and Pomeau, 1986). It utilizes the particle distribution function to describe collective behaviors of fluid molecules. Although promising, the current LBM still have a few shortcomings that limit its general application as a practical computational fluid dynamics tool. One of these shortcomings, which are specifically addressed in this paper, is low order accuracy of the current LBM model.

Generally, there are three types of thermal lattice Boltzmann models have been proposed; multi-speed model (McNamara and Alder, 1993), passive scalar model (Shan, 1997) and double-distribution function (DDF) model (He, Chen and Doolen, 1998). Among these models, the DDF model is reported to be the most stable (Peng, Shu and Chew, 2003) and widely used in simulating
thermal fluid flow problems (Onishi, Chen and Ohahsi, 2001; Peng, Shu and Chew, 2002; Nor Azwadi and Tanahashi, 2007).

In this paper, the DDF thermal LBM is combined with finite difference technique. The finite difference technique is applied to solve the advection term in the governing equations of DDF thermal LBM. This combination contributes in allowing us to increase the accuracy both in time and space. In order to verify the proposed model, a natural convection of air, rectangular enclosure with localized heating from below and symmetrical cooling from the sides is considered.

2. Double-Distribution Function Thermal LBM

The governing equation of the DDF thermal LBM are

\[
\frac{\partial f_i}{\partial t} + c \frac{\partial f_i}{\partial x} = \frac{1}{\tau_f} \left[ f_i - f_i^{eq} \right] + \mathbf{F}
\]

(1)

\[
\frac{\partial g_i}{\partial t} + c \frac{\partial g_i}{\partial x} = \frac{1}{\tau_g} \left[ g_i - g_i^{eq} \right]
\]

(2)

where the density distribution function \( f = f(x,c,t) \) is used to simulate the density and velocity fields and the internal energy density distribution function \( g = g(x,c,t) \) is used to simulate the macroscopic temperature field. \( c \) in Eqs. (1) and (2) is the microscopic velocity and \( \mathbf{F} \) is the external force. Noted that the first term in the right hand side of Eqs. (1) and (2) is the collision term where the BGK approximation (Bhatnagar, Gross and Krook, 1954) has been applied. In this approximation, \( \tau \) is the time to reach equilibrium condition during collision process and known as the time relaxation of distribution function.

Macroscopic density \( \rho \), velocity \( \mathbf{u} \) and temperature \( T \) of the fluid are determined by the following moments of the distribution functions

\[
\rho = \int f^{eq} \, dc \cdot \mathbf{c} \cdot \mathbf{u} = \int c f^{eq} \, dc \cdot \mathbf{u} \quad \text{and} \quad T = \int g^{eq} \, dc
\]

(3)

The equilibrium distribution functions, \( f^{eq} \) and \( g^{eq} \) are chosen such that the continuum macroscopic equations approximated by the governing equation correctly describe the hydrodynamics of the fluid. They are given as

\[
f_i^{eq} = \rho \omega_i \left[ 1 + 3 \mathbf{c}_i \cdot \mathbf{u} + \frac{9}{2} \left( \mathbf{c}_i \cdot \mathbf{u} \right)^2 - \frac{3}{2} \mathbf{u}^2 \right]
\]

(4)

\[
g_i^{eq} = T \omega_i \left[ 1 + 3 \mathbf{c}_i \cdot \mathbf{u} \right]
\]

(5)

The lattice geometry for density and internal energy density distribution function are shown in Fig. 1.

**Figure 1:** (a) Nine-velocity Model, \( \omega_0 = 4/9, \omega_1 = 1/9 \) and \( \omega_5 = 1/36 \) and (b) Four-velocity Model, \( \omega_1 = 1/4 \)
The macroscopic continuity, momentum and energy equations can be obtained from Eqs. (1) and (2) by using multi-scaling expansion procedure (Hou, Zou, Chen and Doolen, 1995). As a result, the viscosity and thermal diffusivity in these models are related to the time relaxations as below

\[ \nu = \frac{\tau_f}{3} \] (6)

\[ \chi = \tau_g \] (7)

### 3. Finite Difference DDF Thermal LBM

In the finite difference formulation of LBM, Eqs. (1) and (2) are once again discretized in phase space and time and solved using one of the several available finite difference numerical schemes. In this paper, the temporal discretization is obtained using second order Runge-Kutta (modified) Euler method. The time evolution of particle distributions is then derived by

\[
\begin{align*}
  f_i^{n+\frac{1}{2}} &= f_i^n + \frac{\Delta t}{2} \left[ -c_i \cdot \nabla f_i^n + \frac{\left( f_i^{eq,n} - f_i^n \right)}{\tau_f} \right] \\
  f_i^{n+1} &= f_i^n + \Delta t \left[ -c_i \cdot \nabla f_i^{n+\frac{1}{2}} + \frac{\left( f_i^{eq,n+\frac{1}{2}} - f_i^{n+\frac{1}{2}} \right)}{\tau_f} \right]
\end{align*}
\] (8) (9)

The third order upwind scheme was applied to calculate the advection term in the evolution of density and temperature equations. From this combination, the accuracy of the finite difference DDF thermal LBM is upgraded to second order in time and third order in space.

### 4. Numerical Results

In this section, the proposed model is applied to simulate the phenomenon of natural convection in a square cavity with localized heating from below and symmetrical cooling from the sides. Symmetrical cooling from the sides is expected to be an efficient cooling option, while partial heating at the lower surface simulates the electronic components such as chips.

Fig. 2 shows a schematic diagram of the setup in the simulation. The lower wall has a various centrally located heat source, \( l = H/5, 2H/5, 3H/5 \) and \( 4H/5 \), which is assumed to be isothermally heated at a constant temperature, \( T_H \).

The Boussinesq approximation is applied to the buoyancy force term. So the external force in Eq. (1) is

\[ F = 3G(c - u)f_i^{eq} \] (10)

The dynamical similarity depends on two dimensionless parameters: the Prandtl number \( Pr \) and the Rayleigh number \( Ra \),

\[ Pr = \frac{\nu}{\chi} \] (11)

\[ Ra = \frac{g \beta \Delta T H^3}{\nu \chi} \] (12)

In all simulations, \( Pr \) is set to be 0.71 in order to simulate air cooling of electronic components.
Figure 2: Geometry and Boundary Conditions of the Problem

Figure 3: Velocity Vectors for Ra = 10^3 and l = H/5

Fig. 3 shows velocity vectors for Ra = 10^3 and l = H/5. They describe that the hot fluid rises above the source until it reaches the top wall, then moves upwards along the horizontal wall before moving downwards along the sidewalls under the effect of cooling. The same phenomenon is observed for every simulated condition.

The main characteristics of natural convection flow are shown in terms of streamlines and isotherm in Figs. 4 to 7. For each case, 30 equally spaced contours have been used to define the corresponding structure.
The flow pattern is characterized by primary flows that rotate clockwise in the right half and counter-clockwise in the left half of the cavity as shown by streamline plots in Fig. 4. Owing to the symmetry, the flow in the left and right halves of the enclosure is identical except for the sense of rotation. Owing to the symmetrical boundary condition, the flow reaches an asymptotic steady state exhibiting a symmetric motion about the vertical centerline of the cavity for all values of $Ra$ and $l$ considered. Due to the same pattern shown by all the simulated conditions, only results obtained for $l = H/5$ are brought as references.

Figs. 5 to 7 show the isotherm plots for each case considered.
At low Rayleigh number, $Ra = 10^3$, the isotherms deviate slightly from a diagonally symmetric structure indicates that the convection intensity is very weak and the main heat transfer mechanism is by conduction. As the Rayliegh number increased, $Ra = 10^4$, the intensity of the recirculation inside the cavity increase, and the cores of the cells move upwards. The effect of convection can be seen from isotherms lines. At $Ra = 10^5$, the formation of thermal boundary layers can be observed, because of increased recirculation intensity. For all values of $Ra$ considered, increasing $l$ for a fixed $Ra$ results in an increase of average Nusselt number (not shown). All of these observations are in good agreement with the results reported in the previous studies (Tsutahara, 1999; Nor Azwadi and Rosdzimin, 2008; Corvaro and Paroncini, 2008).
5. Summary and Concluding Remarks

In this paper, the natural convection in a square cavity with localized heating from below and symmetrical cooling from the sides has been studied using finite difference double-distribution function thermal lattice Boltzmann model. The evolution of lattice Boltzmann equation has been discretised using the third order accuracy finite difference upwind scheme. The flow pattern including vortices and thermal boundary layer can be seen clearly. The results obtained demonstrate that this approach is very efficient procedure to study flow and heat transfer in a differentially heated cavity flow. The extension to 3D computations and high Rayleigh numbers is the subject of further investigations within ongoing research.

Acknowledgment

The author would like to thank Universiti Teknologi Malaysia and Malaysia government for supporting this research activity.
References


