

7th International Conference on Fluid Mechanics, ICFM7

An improved momentum-exchanged immersed boundary-based lattice Boltzmann method for incompressible viscous thermal flows

Mu-Feng Chen^a, Xiao-Dong Niu^{a,*}, Yi-Ren Ma^a, H. Yamaguchi^b, Y. Iwamoto^b^aCollege of Engineering, Shantou University, Shantou 515063, China^bEnergy Conversion Research Center, Doshisha University, Kyoto, Japan

Abstract

An improved momentum-exchanged immersed boundary-based lattice Boltzmann method (MEIB-LBM) is proposed for incompressible viscous flows in this paper. In present work, we come back to the intrinsic feature of LBM, which uses the density distribution function as a dependent variable to evolve the flow field, and use the non-equilibrium density distribution function correction at the neighbour Euler mesh points to satisfy the non-slip boundary condition on the immersed boundary. The improvements for the original MEIB-LBM are that the intrinsic feature of LBM is kept and the flow penetration is removed. Examples, including force convection over a stationary heated circular cylinder for heat flux condition, and natural convection with a buoyant circle particle in viscous fluid, are provided to validate the present method.

© 2015 Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license

(<http://creativecommons.org/licenses/by-nc-nd/4.0/>).

Peer-review under responsibility of The Chinese Society of Theoretical and Applied Mechanics (CSTAM)

Keywords: improved MEIB-LBM, heat transfer, moving boundary;

1. Introduction

In the past years, lattice Boltzmann method (LBM) has received growing attentions for flow problems. For solving boundary condition, the immersed boundary method (IBM) is one of the most promising method currently. IBM is first introduced by Peskin [1] in 1970s to investigate the blood flows problem in the heart.

Niu [2] proposed the idea of momentum-exchange LBM for moving boundary problems. The idea indicates that the fluid-structure interaction is simulated by calculating the difference of the momentum. But, it has been shown by many later researches [3, 4] that there is a flow penetration in immersed boundary.

* Corresponding author. Tel.: 13729201556; fax: 0754-86502941.

E-mail address: xdniu@stu.edu.cn

In this work, we present an improved momentum-exchanged immersed boundary-based lattice Boltzmann method, using the non-equilibrium part of the distribution function to solve the immersed boundary. The same idea of the present work have be applied to the thermal flows. Force convection over a stationary isoflux circular cylinder for heat flux condition, and natural convection with a buoyant circle particle in viscous fluid are tested. The present results show a good agreement with those in the literature.

2. Method

2.1 Lattice Boltzmann method

Lattice Boltzmann Equation coupled double distribution functions model based on Boussinesq approximation can be expressed as:

$$f_{\alpha}(\mathbf{r} + \mathbf{e}_{\alpha}\delta_t, t + \delta_t) = f_{\alpha}(\mathbf{r}, t) - \frac{f_{\alpha}(\mathbf{r}, t) - f_{\alpha}^{eq}(\mathbf{r}, t)}{\tau_f} + \omega_{\alpha}\delta_t(1 - \frac{1}{2\tau_f})\Delta\rho(r_{ij}, t), \quad (1)$$

$$g_{\alpha}(\mathbf{r} + \mathbf{e}_{\alpha}\delta_t, t + \delta_t) = g_{\alpha}(\mathbf{r}, t) - \frac{g_{\alpha}(\mathbf{r}, t) - g_{\alpha}^{eq}(\mathbf{r}, t)}{\tau_g} + \omega_{\alpha}\delta_t(1 - \frac{1}{2\tau_g})\Delta T(r_{ij}, t) \quad (2)$$

where \mathbf{r} is the spatial vector, δ_t denotes the time step, τ_f and τ_g is the dimensionless relaxation time for velocity and temperature, f_{α} and g_{α} is the density and temperature distribution function, f_{α}^{eq} and g_{α}^{eq} is the density and temperature equilibrium distribution function separately, $\Delta\rho(r_{ij}, t)$ and $\Delta T(r_{ij}, t)$ is correction of the non-equilibrium part of the distribution function in Lagrangian points. The equilibrium distribution function can be expressed as

$$f_{\alpha}^{eq}(\mathbf{r}, t) = \omega_{\alpha}(\rho + \rho_0(\frac{3}{c^2}(\mathbf{e}_{\alpha} \cdot \mathbf{u}) + \frac{9}{2c^4}(\mathbf{e}_{\alpha} \cdot \mathbf{u})^2 - \frac{3}{c^2}\mathbf{u}^2)) \quad (3)$$

$$g_{\alpha}^{eq}(\mathbf{r}, t) = \omega_{\alpha}T[1 + \frac{3}{c^2}(\mathbf{e}_{\alpha} \cdot \mathbf{u}) + \frac{9}{2c^4}(\mathbf{e}_{\alpha} \cdot \mathbf{u})^2 - \frac{3}{c^2}\mathbf{u}^2] \quad (4)$$

2.2 Velocity and temperature correction by an improved MEIB-LBM

The boundary configuration is denoted by a series of Lagrangian points, and the flow field covers both internal and external of the object and is represented by a set of Euler mesh points. The interaction between the immersed boundary and the flow can be simulated by calculating the distribution function change generated by boundary onto the flow. The distribution function on boundary points can be approximated by following

$$f_{\alpha}(X_l, t) = \sum_{ij} f_{\alpha}(\mathbf{r}_{ij}, t) D_{ij}(\mathbf{r}_{ij} - \mathbf{X}_l) h^2 \quad \mathbf{g}_{\alpha}(X_l, t) = \sum_{ij} g_{\alpha}(\mathbf{r}_{ij}, t) D_{ij}(\mathbf{r}_{ij} - \mathbf{X}_l) h^2$$

where i, j are the indexes of the Euler mesh points in x and y -directions; $f_{\alpha}(X_l, t)$ and $g_{\alpha}(X_l, t)$ are the distribution function of Lagrangian points on the immersed boundary; $f_{\alpha}(r_{ij}, t)$ and $g_{\alpha}(r_{ij}, t)$ is distribution function in Euler mesh points and h is equal to the space of the mesh. $D_{ij}(r_{ij}, X_l)$ is delta function. The density $\rho(X_l, t)$ and temperature $T(X_l, t)$ of solid boundary can be computed as

$$\rho(X_l, t) = \sum_{\alpha} f_{\alpha}(X_l, t) \quad T(X_l, t) = \sum_{\alpha} g_{\alpha}(X_l, t) + (-\frac{Q(X(S), t)\delta_l}{\rho c_p})$$

To obtain the non-equilibrium part of the distribution function in Lagrangian points, we must substituting boundary condition of velocity \mathbf{u}_b , density $\rho_0(X_l, t)$ and $T(X_l, t)$ temperature into Eqs.(3) and (4) to satisfied the non-slip boundary condition. Bounce back rules is used to obtain a new set of non-equilibrium part of distribution functions on the boundary points and is further projected on the neighbouring Euler points for reflecting the boundary effects as following

$$f_{\alpha, b}^{non-eq}(r_{ij}, t) = \sum_l f_{\alpha}^{non-eq}(X_l, t) D_{ij}(\mathbf{r}_{ij} - \mathbf{X}_l) \Delta s_l, \quad g_{\alpha, b}^{non-eq}(r_{ij}, t) = \sum_l g_{\alpha}^{non-eq}(X_l, t) D_{ij}(\mathbf{r}_{ij} - \mathbf{X}_l) \Delta s_l,$$

where Δs_l is the arc length of the boundary element. The macroscopic density ρ , velocity \mathbf{u} and temperature T can be corrected as

$$\rho\mathbf{u} = \sum_{\alpha} f_{\alpha, ij} \mathbf{e}_{\alpha} + 0.5\delta_t \sum_{\alpha} f_{\alpha, b}^{non-eq}(r_{ij}, t) \mathbf{e}_{\alpha} \quad T(r_{ij}, t) = \sum_{\alpha} g_{\alpha, ij} + 0.5\delta_t \sum_{\alpha} g_{\alpha, b}^{non-eq}(r_{ij}, t)$$

The important advantage of this method is that it avoid to introduce the forcing term and heat source into the

momentum equations, which is generated by the boundary onto the fluid. Instead, the non-equilibrium part of the distribution function in Lagrangian points is calculated to obtain the macro quantity near the boundary.

3. Numerical results and discussions

3.1 Heat flux condition for forced convection over a stationary isoflux circular cylinder

In this section, forced convection over a stationary circular cylinder with a constant heat flux boundary condition is considered. The flow behaviour and isotherm patterns is related to the Reynolds number $Re = \rho u_{\infty} D / \mu$ and the Prandtl number $Pr = \mu c_p / k$, where ρ , u_{∞} , and D means the fluid density, free stream velocity, and cylinder diameter respectively, μ , c_p and k is the dynamic viscosity, specified heat and the thermal diffusivity, respectively. In present study, numerical investigations are carried out for $Re=20$, 40, and $Pr = 0.7$. Drag coefficient Cd , streamlines and isotherms ,average Nusselt number Nu on the boundary are listed.

The domain size is $40D \times 30D$ channel, in middle of which stands stationary a circular cylinder with a constant heat flux condition $-\partial T / \partial y = 1.0$, and $\partial u / \partial y = \partial T / \partial y = 0$ is defined for the top and bottom wall. The cylinder is $15D$ far away from the inlet, which has $\mathbf{u} = (u_{\infty}, 0)$ and $T = 0$. At the outlet, the homogeneous Neumann boundary condition is applied, which is $\partial u / \partial x = \partial T / \partial x = 0$.

Figure 1 shows that penetration of streamlines does not happen near the boundary, meaning that the non-slip boundary condition is well satisfied. The outcome indicated that present method improves the accuracy of the boundary. The drag coefficient Cd is listed in Table 1, agreeing well with the literature.

Figure 2 shows the isotherms nearby the cylinder boundary for $Re=20$ and 40. It is obviously to found that the isotherms in the front surface of the cylinder is dense, while the rear surface is relatively sparse. Table 2 shows the average Nu on the cylinder surface, appearing a good agreement with reference data.

Table 1 Comparison of the Cd at $Re=20$ and 40

References	$Re=20$	$Re=40$
Ren et al. [5]	2.126	1.568
Niu et al. [2]	2.144	1.589
Hu et al. [3]	2.213	1.660
Present	2.261	1.647

Table 2 comparison of average Nu at $Re=20$ and 40

References	$Re=20$	$Re=40$
Ren et al.[5]	2.7413	3.7407
Ahmad et al.[4]	2.6620	3.4720
Dhiman et al.[6]	2.8630	3.7930
Present	2.8414	3.6872

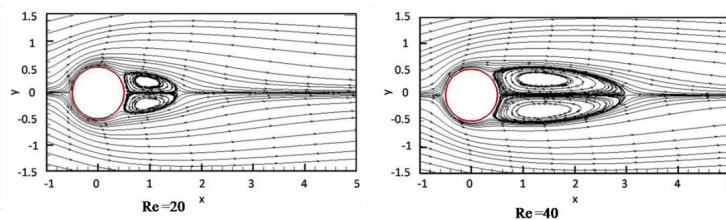


Fig 1. Streamlines at $Re=20$ and 40.

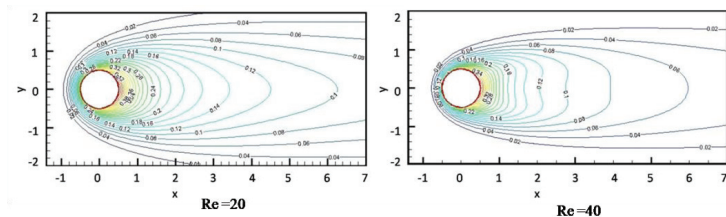


Fig 2. Isothermal at $Re=20$ and 40.

3.2 Natural convection with a neutrally buoyant particle in viscous fluid for moving boundary

Natural convection has been investigated by many researchers [7-9]. We extended it to investigate the case in a square cavity of natural convection with a suspended circle particle in viscous fluid.

Figure 3 shows the computational domain, boundary condition and walls conditions. The grid is taken as 200×200 . In the position of (0.5,0.9) stands a neutrally buoyant particle, with the isoflux condition of $-\partial T/\partial n=5.0$, density of $\rho=1.005$, and diameter of $d=0.05$. For natural convection, the flow behavior is related to the Prandtl number $Pr=\nu/\chi$ and Rayleigh number $Ra=g\beta\Delta TPrh^3/\nu^2$. Where ρ , ν , χ , g , β , and h means the fluid density, kinematic viscosity, thermal diffusivity, acceleration of gravity, thermal expansion coefficient, reference length respectively, ΔT is the difference of high and low temperature wall. In present case, $Pr=0.71$. The Boussinesq approximation is applied.

Figures 4 and 5 show the streamlines and isotherms when the flow reach a steady state at $Ra=10^3, 10^4, 10^5$. These plots agree well with those in referent literature.

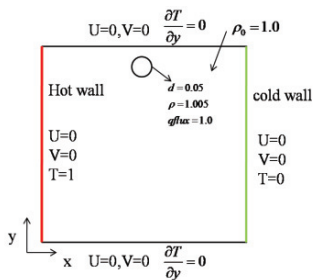


Fig 3 computational domain conditions

Table 3 Nu of cold wall at $Ra=20, 40$ with a particle

References	10^3	10^4	10^5
Peng et al. [7]	1.117	2.241	4.511
Wang et al. [8]	1.115	2.232	4.491
Shu et al.[9]	1.118	2.245	4.523
present	1.268	2.416	4.551

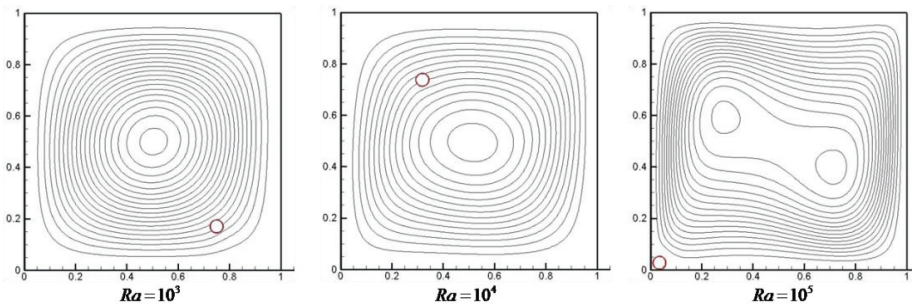


Fig 4. Streamlines at Rayleigh number of $Ra=10^3, 10^4$ and 10^5

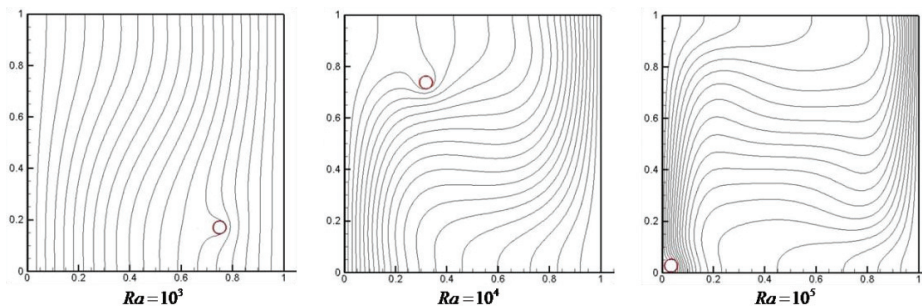


Fig 5. Isothermal at Rayleigh number of $Ra=10^3, 10^4$ and 10^5

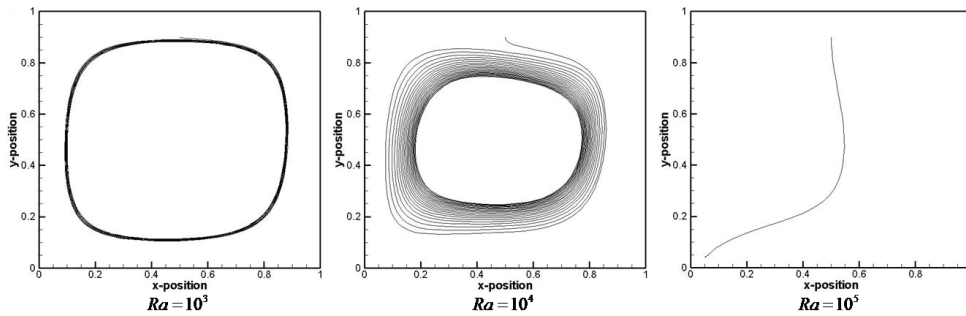


Fig 6. Position trajectory at Rayleigh number of $Ra=10^3, 10^4$ and 10^5

Table 3 lists the Nusselt number at $Ra=10^3, 10^4, 10^5$, showing good agree with those in referent literature [7-9]. It is easy to found that Nu become greater as the increase of Ra . Compare the Nu in the same Ra , the Nu improve in present method, indicating that the heated particle can improve heat transfer characteristic of the fluid.

The particle has a different track as the change of Rayleigh number, which can be observed in Fig 6. It can be seen that when the $Ra=10^3$, the trajectory of the particle is nearly a circular. However, radius of position trajectory decrease slowly with time at $Ra=10^4$. When the $Ra=10^5$, the radius in the first circle time become even small, and then bigger, and finally fall down in the bottom left of the cavity.

4. Conclusion

An improved momentum-exchanged immersed boundary-based lattice Boltzmann method for incompressible viscous thermal flows is presented. We come back to the intrinsic feature of LBM, using the density distribution function as a dependent variable to evolve the flow field. This method have applied to simulated static boundary of heat flux condition for forced convection over a stationary isoflux circular cylinder, and moving boundary of natural convection with a buoyant circle particle in viscous fluid. Note that the results validate its capability and efficiency to simulation of incompressible viscous thermal flows.

Acknowledgements

This work was supported by NSFC Project Grant No. 1137168.

References

- [1] C.S. Peskin, Numerical analysis of blood flow in the heart, *J. Comput. Phys.* 25 (1977) 220–252.
- [2] X.D. Niu, C. Shu, A momentum exchange-based immersed boundary-lattice Boltzmann method for simulating incompressible viscous flows, *Physics Letters A* 354 (2006) 173–182.
- [3] Yang Hu, Haizhuan Yuan, An improved momentum exchanged-based immersed boundary-lattice Boltzmann method by using an iterative technique, *Computers and Mathematics with Applications* 68 (2014) 140–155.
- [4] R.A. Ahmad, Z.H. Qureshi, Laminar mixed convection from a uniform heat flux horizontal cylinder in a crossflow, *J. Thermophys. Heat Transfer* 6 (1992) 277–287.
- [5] W.W. Ren, C. Shu, An efficient immersed boundary method for thermal flow problems with heat flux boundary conditions, *Int. J. of Heat and Mass Transfer* 64 (2013) 694–705.
- [6] A.K. Dhiman, R.P. Chhabra, A. Sharma, V. Eswaran, Effects of Reynolds and Prandtl numbers on the heat transfer across a square cylinder in the steady flow regime, *Numer. Heat Transfer* 49 (2006) 717–731.
- [7] Peng Y, Shu C, Chew YT. Simplified thermal lattice Boltzmann model for incompressible thermal flows. *Phys Rev E* 2003;68:02671–2678.
- [8] Y. Wang, C. Shu, Thermal lattice Boltzmann flux solver and its application for simulation of incompressible thermal flows. *Computers & Fluids* 94 (2014) 98–111.
- [9] Shu C, Xue H. Comparison of two approaches for implementing stream function boundary condition in DQ simulation of natural in a square cavity. *Int J Heat Fluid Flow* 19(1998) 59–68.