ABSTRACT

A novel thermal Lattice Boltzmann Method is introduced in which convection heat transfer from a solid particle surface is simulated using the Lattice Boltzmann Method. Fluid-solid interaction and its relevant heat transfer is simulated using an external boundary force method in which momentum and heat exchange at the interface is obtained by interpolation functions. This method is capable of simulating heat transfer from a particle in a fluid flow as well as particulate flows with many particles. It is also a computationally efficient method as all computations are performed on a regular uniform lattice. Results of local and averaged Nusselt number obtained for convection heat transfer from a stationary circular cylinder in a uniform flow as well as natural convection in an enclosure demonstrate the accuracy of the method compared to previous published works.

KEYWORDS: Lattice Boltzmann Method, External Boundary force, natural convection

INTRODUCTION

Analysis of heat transfer in a particulate flow is a challenging problem mainly due to the complex mechanism of momentum and heat exchange at the interface between solid and fluid. While various CFD methods have been used in simulating particulate flows, the Lattice Boltzmann Method (LBM) has attracted specific attention mainly due to its computational efficiency [1]. In spite of considerable progress in modelling the fluid flow aspect of particulate flows, simulations of heat transfer with thermal LBM have been limited.

Simulation of heat transfer using LBM was first performed successfully using a double distribution function model proposed by He, et. al. [2]. They proved that such a double distribution function leads to the correct derivation of the energy equation for an incompressible flow. Wang, et. al. [3] presented a LBM algorithm for fluid-solid conjugate heat transfer. They used a new generalized heat generation implementation which ensures temperature and heat flux continuity at the interface and tested their method in a micro-channel flow.

One of the main challenges in simulation of particulate flow is that solid-fluid interaction force plays an essential role in predicting correct momentum and heat transfer between solid and fluid. Goldstein et al. [4] proposed a method, called External Boundary Force (EBF), to obtain solid-fluid interaction forces. Their method which is similar to the Immersed Boundary Method (IBM) [5] uses an Eulerian grid for fluid and a Lagrangian one for the motion of particles which could be discretized with a boundary-fitted grid. Both standard IBM and EBF methods are second order accurate. The main difference between IBM and EBF is in the representation of motion of the solid body: the main idea of IBM is to use the physical boundary as a spring with high stiffness between internal solid nodes and calculate the acceleration or particle from the energy equation while the EBF method obtains acceleration from the dynamics of rigid body. To implement the no-slip condition at a particle surface, Wu and Aidun [6] added a body force term to the LB equation which is obtained from momentum difference between interpolated fluid velocity at solid boundary nodes and solid velocity at the same nodes, as a source term. They showed that EBF method is a reliable method for both deformable and non-deformable solid particle motion simulation in a fluid. They implemented the effect of fluid-solid interaction force by adding an additional term to the collision step in Boltzmann equation.

To the best of the authors’ knowledge, the external boundary method has not been extended to heat transfer problems. The present paper presents the Thermal External Boundary Method (TEBM) and its application to forced and natural convection simulations.

EXTERNAL BOUNDARY FORCE

The External Boundary Force (EBF) is a method for simulating fluid-solid interaction force based on momentum exchange at the interface. Considering two sets of grid points, namely fluid Eulerian and solid Lagrangian ones, (cf. Fig. (1)) we note that solid nodes do not correspond to the fluid nodes in general, however, both fluid and solid velocities at boundary nodes need to be computed. Knowing these values, momentum exchange between fluid and solid can be computed. The fluid velocity at solid boundary nodes is obtained using an interpolation function which is explained in details below.

Figure 1 shows part of a computational domain consisting of a solid circular particle submerged in a fluid domain. A series
of Lagrangian nodes located on the interface between solid and fluid are superimposed on the uniform Eulerian grids. The set of points are shown as $x^b$ and $x^f$ representing position vectors of solid nodes located on the boundary of particle and fluid nodes respectively. 

$$\delta(x) = \begin{cases} \frac{1}{(2\pi h)^3} & \text{if } |x^b - x^f| < 2h \\ 0 & \text{otherwise} \end{cases}$$ \hspace{1cm} (2)

Here ‘$h$’ is the unit lattice length. The accuracy of this function is of first order. The above relations are written for the present time. Knowing fluid velocity at boundary nodes in previous time step, it is possible to compute rate of momentum change and the subsequent fluid-solid interaction force. In EBF, fluid velocity at a solid boundary node is considered to be equal to the velocity of solid boundary nodes at previous time step. It is worth mentioning that solid particle velocity is a known quantity at previous time step which starts from known initial conditions. It can be expressed as:

$$U^f(x^b, t - \Delta t^{lbm}) = U^s(x^b, t - \Delta t^{lbm})$$ \hspace{1cm} (3)

where $\Delta t^{lbm}$ and $U^s$ are Lattice Boltzmann time step (which is considered as 1) and particle velocity, respectively. The fluid-solid interaction force $F_{fsi}$ acting on solid boundary nodes is obtained from rate of momentum exchange as:

$$F_{fsi}(x^b, t) = \rho_f[U^f(x^b, t) - U^f(x^b, t - \Delta t^{lbm})] / \Delta t^{lbm}$$ \hspace{1cm} (4)

where $\rho_f$ is the density of fluid. The next step is to find interaction force at fluid nodes adjacent to solid boundary which is obtained through similar interpolation using Dirac delta function with a negative sign due to Newton’s third law:

$$\mathbf{g}(x^f, t) = - \sum_{k=1}^{m} F_{fsi}(x^b_k, t) \mathbf{D}(x^b_k - x^f)$$ \hspace{1cm} (5)

“$\mathbf{g}$” is introduced as External Boundary Force and will be used as source term in Lattice Boltzmann equation. Except fluid-solid interaction force, there might be other external forces such as gravitational, lubrication, buoyancy and magnetic forces. The total force acting on solid boundary nodes expressed as:

$$\mathbf{F}(x^b, t) = F_{fsi}(x^b, t) + \mathbf{F}_{ext}(x^b, t)$$ \hspace{1cm} (6)

The force $\mathbf{F}(x^b, t)$ in equation 6 exerts on solid boundary nodes, while the total force and torque act on immersed particle boundary should be obtained as:

$$\mathbf{F}^T = \sum_{a=1}^{N} F(x^b_a, t)$$ \hspace{1cm} (7)

$$\mathbf{T}^T = \sum_{a=1}^{N} \left[ F(x^b_a, t) \right] \left( \mathbf{x}^{cg} - x^b_a \right)$$ \hspace{1cm} (8)

Here “$N$” is the number of nodes on the boundary, “$\mathbf{x}^{cg}$” is the center of gravity of the particle. The Newtonian dynamics equations for a solid body are expressed as:

$$\frac{d\mathbf{M}}{dt} = \mathbf{F}^T$$ \hspace{1cm} (9)

$$\mathbf{I} \frac{d\Omega}{dt} + \mathbf{\Omega} \times \left( \mathbf{I} \mathbf{\Omega} \right) = \mathbf{T}^T$$ \hspace{1cm} (10)

where “$\mathbf{M}$”, “$\mathbf{T}$”, “$\mathbf{\Omega}$” and “$\mathbf{U}$” are the mass, inertial tensor of the particle, angular and linear velocities of the solid body, respectively. Fourth-order Runge-Kutta method could be used to solve the equation (9) and (10) to obtain the linear and angular velocities of the particle.

**THERMAL EXTERNAL BOUNDARY METHOD**

Building on the discussion in the previous section, the thermal external boundary method is based on obtaining fluid temperature, $T^f$, at solid boundary nodes through interpolation using Dirac delta function used in equation (2), i.e.:

$$T^f_a = \sum_{k=1}^{n} T^f_k \mathbf{D}(x^b_k - x^f_a)$$ \hspace{1cm} (11)

Here, $T^f_k$ is the fluid node temperature near the boundary node. Temperature of fluid nodes on solid boundary at previous time step are defined as:

$$T^s(x^b, t - \Delta t^{lbm}) = T^s(x^b, t - \Delta t^{lbm})$$ \hspace{1cm} (12)

where $T^s$ is the temperature of solid boundary nodes which is known. The heat flux on the surface of solid particle is obtained from rate of change of temperature at a fluid node corresponds to a solid boundary point as:

$$Q_{fsi}(x^b, t) = \rho_c C_p \left[ T^f(x^b, t) - T^s(x^b, t - \Delta t^{lbm}) \right] / \Delta t^{lbm}$$ \hspace{1cm} (13)

This parameter is considered a heat source/sink term in temperature evolution equation as discussed later, but it has to be obtained for fluid nodes through interpolation. Therefore, heat source/sink for fluid neighboring-nodes is obtained as:

$$Q(x^f_a, t) = - \sum_{k=1}^{m} Q_{fsi}(x^b_k, t) \mathbf{D}(x^b_k - x^f_a)$$ \hspace{1cm} (14)
where \( Q(x',t) \) which is defined as Thermal External Boundary source, is added to Thermal Lattice Boltzmann Equation as follows:

\[
T_a(x' + c_a \Delta t, t + \Delta t) - T_a(x', t) = -\frac{T_a(x',t) - T^e_q(x',t)}{\tau} + \Delta t Q_a
\]

The above equation is an evolution equation for the temperature field analogous to the Lattice Boltzmann equation which is expressed in the form:

\[
f_a(x' + c_a \Delta t, t + \Delta t) - f_a(x', t) =
-\frac{f_a(x',t) - T^e_q(x',t)}{\tau} + \Delta t F_a
\]

(16)

\( T_a \) is temperature distribution function in Eq. (15), \( T^e_q \) is the equilibrium distribution function used for direction “a” and “\( \tau \)” is thermal relaxation time. In order to solve the incompressible thermal flow, we substitute \( \rho = \rho_0 \) into the equilibrium distribution function as proposed by Peng [7]:

\[
T^e_q = \frac{\rho_0 T_0 u u}{3} \frac{\gamma}{c^2} \] (17a)

\[
T^e_{1-\alpha} = \frac{\rho_0 T}{\delta x} \left[ \frac{3c_a u}{2c^2} + \frac{g(c_a u)^2}{2c^4} - \frac{3 u u}{2c^2} \right] \] (17b)

\[
T^e_{s=\alpha} = \frac{\rho_0 T}{\delta x} \left[ 3 + \frac{6c_a u}{c^2} + \frac{g(c_a u)^2}{2c^4} - \frac{3 u u}{2c^2} \right] \] (17c)

thermal diffusivity “\( \chi \)” and macroscopic fluid temperature are calculated using:

\[
\chi = 2(\tau - 0.5) c_d^2 \Delta t
\]

(18)

\[
T(x', t) = \frac{1}{\rho_0} \sum_a T_a(x', t)
\]

(19)

RESULTS

As a first validation test case, forced convective heat transfer from a heated circular cylinder placed in a uniform flow is studied. Figure 2 shows this flow geometry which consists of a cylinder with diameter of D and its computational domain of 30D\times20D. The cylinder is placed at a distance of 8D downstream of inlet boundary. This test case has been studied by other researchers using different numerical methods [7].

Two important parameters in this problem are the Prandtl number, \( Pr \), and Reynolds number, \( Re \). The aim of this simulation is to obtain local and mean Nusselt numbers on the surface of the cylinder as a function of Reynolds number and Prandtl number. The local and mean Nusselt numbers are expressed as:

\[
Nu = \frac{\partial T}{\partial n} \left|_{wall} \right.
\]

(32)

\[
\overline{Nu} = \frac{1}{S} \int_S \overline{Nu} \, ds
\]

(33)

where “\( S \)” is the circumferential length of the cylinder surface. Local temperature gradient is computed by using second-order approximation as:

\[
\frac{\partial T}{\partial n} \approx -3d_\theta + 4\theta(r+\delta x, \theta) - 3d_\theta(r+2\delta x, \theta)
\]

(34)

Simulations were carried out for a regular rectangular domain of dimensions 30D\times20D. Hydrodynamic boundary conditions used on top and bottom boundaries were free slip boundary conditions. Boundary conditions on inlet and outlet boundaries were applied based on known velocity \( U = 1 \) and zero velocity gradient, respectively. Thermal boundary conditions consist of adiabatic boundary condition for top and bottom walls, constant inlet temperature and zero temperature gradient for outlet boundary.

Simulations were done for three different Reynolds numbers of \( Re=10, 20, 40 \), all at a fixed Prandtl number of \( Pr = 0.7 \). The drag coefficient is defined as:

\[
C_d = \frac{F_D}{2 \rho U^2 D}
\]

(35)

where \( F_D \) is the drag force computed from the fluid-solid interaction force. The drag coefficients obtained for \( Re=10, 20 \) and 30 are 1.863, 2.44 and 3.381 respectively which show an error of less than 2\% compared to previous published data of Ref. [8]. Figure 3 shows temperature contours for uniform flow on a circular cylinder at \( Re=20 \). Higher temperature gradient is observed upstream of the cylinder compared to its downstream region which covers the recirculating flow region.

Figure 3. Temperature contours for flow over a circular cylinder at \( Re=20 \).

The second test case considers natural convection between a hot cylinder and a cold square enclosure. Here a cylinder with a diameter of 80 lattice units is placed at the center of a square domain consisting of 201\times201 lattice nodes. The nob-dimensional temperature of the cylinder is prescribed as 1 while that of the square enclosure is 0. The computations are
performed at a Rayleigh number of Ra=$10^4$. Figure 4 shows streamlines and temperature contours for this geometry.

![Streamlines and temperature contours](image)

Figure 4. (a) Streamlines and (b) temperature contours for natural convection between a cylinder and a square enclosure.

Results of mean Nusselt number obtained for different cylinder diameter at three Rayleigh numbers of $10^4$, $10^5$ and $10^6$, are compared with previous published work in table 1.

Table 1. Mean Nusselt number values for natural convection between cylinder and square enclosure.

<table>
<thead>
<tr>
<th>Ra</th>
<th>R/L</th>
<th>Present</th>
<th>Seta[8]</th>
</tr>
</thead>
<tbody>
<tr>
<td>$10^4$</td>
<td>0.1</td>
<td>2.296</td>
<td>2.0206</td>
</tr>
<tr>
<td>$10^5$</td>
<td>0.2</td>
<td>5.461</td>
<td>5.253</td>
</tr>
<tr>
<td>$10^6$</td>
<td>0.3</td>
<td>13.128</td>
<td>12.87</td>
</tr>
</tbody>
</table>

CONCLUSIONS

The Thermal External Boundary Method which is an extension of the External Boundary Force method in conjunction with the Lattice Boltzmann method, was shown to be capable of predicting temperature distribution and heat flux in natural and forced convection in simple geometries consisting of uniform flow over a circular cylinder and natural convection in a square enclosure from a heated cylinder placed at its center. The computed Nusselt numbers for these two geometries show good correspondence with previous published work. The TEBM formulation should provide a good and efficient computational platform for simulating convective heat transfer from a moving particle in single or multiple particle problems to be investigated in the future.

REFERENCES


