Computers and Fluids 162 (2018) 152–159

Contents lists available at ScienceDirect

Computers and Fluids

journal homepage: www.elsevier.com/locate/compfluid

Boundary conditions for lattice Boltzmann method with multispeed lattices

H.C. Lee [a,b,*], S. Bawazeer [a], A.A. Mohamad [a]

[a] Department of Mechanical and Manufacturing Engineering, The University of Calgary, Calgary, Alberta T2N 1N4, Canada
[b] Department of Mechanics and Aerospace Engineering, Southern University of Science and Technology, Shenzhen, Guangdong 518055, China

ARTICLE INFO

Article history:
Received 16 February 2017
Revised 6 December 2017
Accepted 21 December 2017
Available online 22 December 2017

Keywords:
Lattice Boltzmann method
Multispeed lattices
On-site velocity boundary conditions
D2Q17

ABSTRACT

Boundary methods and boundary node treatments for a multispeed lattice are presented in detail for incompressible isothermal flows, which remain a challenging application of the lattice Boltzmann method. We developed a completely generic way of handling boundaries with known velocity for multispeed lattices by extending and improving the on-site boundary conditions proposed by Hecht et al. [1], which was only intended for single speed lattice. In addition, we studied two ways of treating the unknown distribution functions that span several layers of nodes, namely, the external treatment (the inner layer is the boundary) and the internal treatment (the outermost layer is the boundary). The external treatment requires the use of ghost nodes at the boundary, whereas the internal treatment relies on an interpolation technique. The external treatment is found to offer more stability than the internal treatment when an open boundary condition is involved. The improved on-site boundary method for multispeed lattices is shown to be local, second-order accurate. Numerical validation is demonstrated with classical benchmark problems, namely, lid-driven square cavity, channel, and back-step flows, at different Reynolds numbers and grid resolutions.

1. Introduction

The rapid increase in computational power has attracted considerable interest in the lattice Boltzmann model with more discrete velocities (multispeed lattices), where the number of discrete velocities can be as high as 125 in three dimensions [2]. This enables correct recovery of the Fourier–Navier–Stokes equations in the hydrodynamic limit for thermal flows [3] and the study of flows beyond the Navier–Stokes hydrodynamics [4]. However, the complexity of modeling the boundary conditions accurately for multispeed lattices has limited its widespread use because the distribution functions occupying not only the boundary itself but also nodes adjacent to the boundary are unknown. Therefore, it is crucial to determine the appropriate equations and treatments when computing the unknown distribution functions, which span several layers of the computational domain for a given boundary condition. In the literature, only two boundary treatment methodologies have been proposed for multispeed lattices: the kinetic-diffuse boundary condition [5] and the general regularized boundary condition (GRBC) [6]. The former is more suitable for microflow applications, whereas the latter is applicable for most flow types on any lattice topology. However, the GRBC requires a significant number of computations, as it requires an overdetermined matrix to be solved at each boundary node. Hence, a simple boundary method for multispeed lattices is of particular interest.

In this paper, we propose a simple yet powerful boundary method for 2D multispeed lattices that can be easily extended to 3D multispeed lattices. The proposed boundary method is based on the work of Hecht et al. [1], where the on-site velocity boundary method was applied to a D3Q19 lattice. Although the boundary method in Ref. [1]. was implemented for D3Q19, the resulting formula cannot be employed directly for multispeed lattices because the distribution functions adjacent to the boundary nodes are missing. Thus, the on-site velocity boundary method is extended and further improved for multispeed lattices in this work to yield a more stable boundary method.

For multispeed lattices, it is necessary to distinguish which layers of nodes belong to the boundary and which belong to the fluid. This is because the unknown distribution functions span several layers of nodes instead of occurring just at the boundary itself for standard lattices (D2Q9 or D3Q19). Two approaches are currently used to treat multilayer lattices, namely, the external treatment, where layers of nodes are grouped to represent identical boundary conditions, or the internal treatment, where the outer layer of

* Corresponding author at: Department of Mechanical and Manufacturing Engineering, The University of Calgary, Calgary, Alberta T2N 1N4, Canada.
E-mail address: hsulee@ucalgary.ca (H.C. Lee).

https://doi.org/10.1016/j.compfluid.2017.12.011
0045-7930/© 2017 Published by Elsevier Ltd.

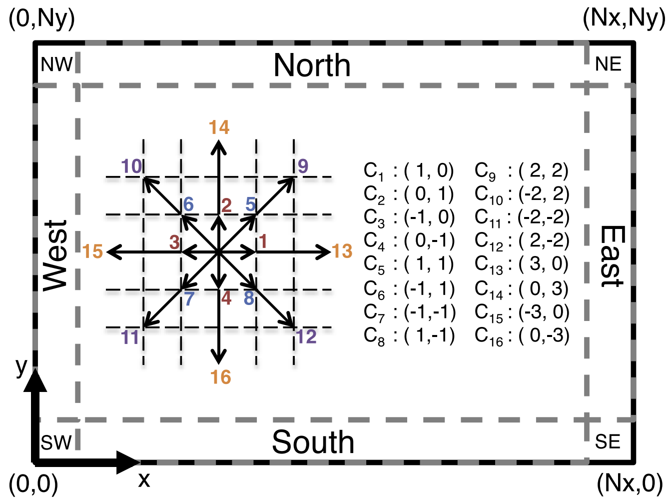

Fig. 1. Illustration of the computational domain used in the first two benchmark cases. The D2Q17 lattice and its corresponding microscopic velocity set (c_i) employed in this study are shown inside the computational domain. The weights (w_i) are: $w_0 = (575 + 193\sqrt{193})/8100$, $w_{1-4} = (3355 - 91\sqrt{193})/18000$, $w_{5-8} = (655 + 17\sqrt{193})/27000$, $w_{9-12} = (685 - 49\sqrt{193})/54000$, and $w_{13-16} = (1445 - 101\sqrt{193})/162000$.

nodes represents the boundary condition and the inner layers belong to the fluid domain. However, there are currently no insightful discussions into which treatment is suitable or more favorable for multispeed lattices. Therefore, we will also discuss in detail how to treat multispeed boundaries externally and internally, as well as the advantages and disadvantages of each type of treatment.

The article is organized as follows. In Section 2, the lattice Boltzmann method (LBM) is briefly described, and Section 3 explains the two types of boundary treatment for multispeed lattices. Section 4 provides a detailed explanation and derivation of the extension and improvement of the existing on-site velocity boundary for multispeed lattices. Numerical validations of the proposed boundary method and treatments are presented in Section 5, and conclusions are drawn in Section 6.

2. High-order lattice Boltzmann model

The LBM scheme adopted in this study is the widely used single-relaxation-time lattice Bhatnagar–Gross–Krook (LBGK) model:

$$f_i(x + c_i, t + 1) - f_i(x, t) = \omega(f_i^{eq}(x, t) - f_i(x, t)) \tag{1}$$

where f_i is the discrete probability distribution function for finding a particle with velocity c_i at position x and time t. $\omega \left(\frac{2c_s^2}{2\nu + c_s^2}\right)$ is the relaxation parameter, f_i^{eq} is the discrete Maxwell–Boltzmann distribution, and c_s is the speed of sound. Multiscale Chapman–Enskog expansion will yield the Navier–Stokes equations with a kinematic viscosity of $\nu = c_s^2(1/\omega - 1/2)$; readers are directed to [7,8] for more details regarding the derivations. In this paper, we employed D2Q17, where the weights w and speed of sound c_s can be found in [5], and the vectors representing the microscopic velocity set c_i are shown in Fig. 1. D2Q17 was chosen mainly because the weights and velocity are optimally derived from the Hermite polynomial [4]. Note that the boundary treatments described in this paper can be easily extended to D2Q25 or D3Q125. The equilibrium distribution up to third order for isothermal flows is used in this study; it is given by

$$f_i^{eq} = w_i\rho\left\{1 + \frac{c_{i,\alpha}\nu_\alpha}{c_s^2} + \frac{1}{2}\left[\frac{(c_{i,\alpha}\nu_\alpha)^2}{c_s^4} - \frac{\nu_\alpha\nu_\alpha}{c_s^2}\right] + \frac{c_{i,\alpha}\nu_\alpha}{6c_s^2}\left[\frac{(c_{i,\alpha}\nu_\alpha)^2}{c_s^4} - 3\frac{\nu_\alpha\nu_\alpha}{c_s^2}\right]\right\} \tag{2}$$

where ρ is the density, and ν_α is the macroscopic velocity component. The LBM algorithm used in this study is summarized in four main steps:

- Collision step: Populations are relaxed by the rule

  $f_i(x, t + 1) = f_i(x, t) + \omega\big(f_i^{eq}(x, t) - f_i(x, t)\big)$
- Streaming step: Populations are displaced in the direction corresponding to c_i:

  $f_i(x, t + 1) \rightarrow f_i(x + c_i, t + 1))$
- Boundary conditions: The missing distribution functions are replaced according to the treatment of the extra layers.
- The macroscopic values (ρ, ν_x, and ν_y) are computed using Eqs. (4)–(6), which are shown in Section 4.

The LBM-LBGK scheme is described in detail in [9,10].

3. Boundary node treatments

A unique feature of multispeed lattices is that the unknown distribution functions span several layer of nodes, as depicted in Fig. 2. Hence, it is crucial to distinguish which nodes belong to the boundary and which nodes belong to the fluid to ensure that the intended physics is solved. There are two ways to differentiate between the boundary and fluid nodes, namely, the external and internal treatments. In the external treatment, nodes across several layers are grouped together as Group I, II, or III (see Fig. 2(a)). The nodes in each group are subjected to identical macroscopic values, boundary conditions, and unknown distribution functions. The grouped nodes share an unknown distribution similar to that of the nodes with the most unknowns, i.e., $l_x = 0$ for group II. For corner nodes (Group III), the unknown distribution functions for nodes located at ($l_x = 1$, $l_y = 1$) and ($l_x = 2$, $l_y = 2$) will be identical to those of the node located at ($l_x = 0, l_y = 0$), which contains the most unknown distribution functions. Therefore, the external treatment requires additional layers of nodes for each boundary condition, and the fluid nodes consist of known distribution functions.

The internal treatment, on the other hand, shares the same methodology as those standard lattice models in which the boundary condition is applied only at the nodes located in the first layer of the computational domain, as shown in Fig. 2(b). However, the macroscopic values for the fluid nodes located along the second and third layers adjacent to the boundary cannot be obtained using Eqs. (4)–(6) for multispeed models because several of the distribution functions remain unknown even after the streaming process. Therefore, we need to interpolate the macroscopic values from the boundary and also from the nodes located in the fourth layers ($l_x = 3$) to reconstruct the unknown distribution functions for the fluid nodes located in the second and third layers. For example, the velocity components for the fluid nodes located along $l_x = 1$ and $l_x = 2$, as shown in Fig. 2(b), are interpolated through the following relation:

$$W_{l_x} = W_{l_x=0} + l_x(W_{l_x=3} - W_{l_x=0})/3 \tag{3}$$

where W represents the macroscopic x and y velocity components. For the fluid nodes adjacent to the corner node (labeled as group IV in Fig. 2(b)), the interpolation involves the use of four nodes