International Journal of Heat and Mass Transfer 127 (2018) 107–115

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Numerical prediction of thin liquid film near the solid wall for hydraulic cavitating flow in microchannel by a multiphase lattice Boltzmann model

Jun Cai [a,b,*], Xiulan Huai [a,b], Bin Liu [a], Zhendong Cui [b]

[a] Institute of Engineering Thermophysics, Chinese Academy of Sciences, Beijing 100190, China
[b] University of Chinese Academy of Sciences, Beijing 100080, China

A R T I C L E   I N F O

Article history:
Received 2 November 2017
Received in revised form 10 April 2018
Accepted 28 June 2018

Keywords:
Hydrodynamic cavitation
Cavitating flow
Thin liquid film
Lattice Boltzmann
Microchannel

A B S T R A C T

Based on a multiphase lattice Boltzmann (LB) model integrating body forces such as inter-particle interaction force and fluid-solid interaction force, a numerical prediction of thin liquid film near the solid wall for hydraulic cavitating flow in microchannel is conducted in the present work. The validity of model is tested by means of Laplace law and experimental result. On this basis, a full thin liquid film near the solid wall is successfully predicted, which proves that the single-component multiphase LB model integrated with various interaction forces is a good solution to this type of problem like hydraulic cavitating flow in microchannel. The further simulation indicates that the fluid-solid interaction strength (g_s) has a significant effect on the formation of thin liquid film. A critical value ($g_s = -3.5$) of fluid-solid interaction strength is found, and thin liquid film fails to be predicted at $g_s > -3.5$. A simple simulation on the contact angle (θ) of droplet on the solid surface is conducted to analyze the reason for this phenomenon. It is found that a full thin liquid film could be successfully predicted only when the solid surface is fully wetted by liquid, i.e., $\theta = 0°$ corresponding to $g_s \leq -3.5$. The current research provides a potential way for future investigation on heat transfer accompanied by hydraulic cavitating flow.

1. Introduction

Cavitation is a pressure-related process and occurs when local pressure falls below vapor pressure corresponding to liquid temperature. In case of acoustic cavitation, the local pressure fluctuation caused by acoustic wave is responsible for cavitation phenomenon [1], while hydrodynamic cavitation occurs when liquid passes through a contraction-diverging device such as throttling valve, orifice plate and venturi [2]. Although cavitation phenomenon was first found due to the negative effect (cavitation erosion), its positive significance is gradually recognized with the deepening of research. Based on various cavitation effects (high temperature, high pressure, high-speed micro-jet) [3,4], cavitation can be applied to such fields as cleaning, cutting, drinking water disinfection [5], and organic wastewater degradation [6,7]. In recent years, hydrodynamic cavitation was proved to be able to enhance liquid convection heat transfer in microchannels [8–10]. Therefore, more and more attention has been paid to this field. When studying heat transfer accompanied by cavitating flow in micochannel, it is very important to accurately simulate the status of flow field for obtaining the reasonable results.

So far, the commonly-used cavitation model includes full cavitation model (FCM) [11], Schner-Sauer model [12], and Zwart-Gerber-Belamri model [13]. These models have been embedded into the commercial CFD softwares, e.g., Ansys Fluent and CFX, and play an important role in cavitation simulation. Nevertheless, because they are based on the assumption of continuous medium, these models are incompatible with the interface tracking algorithm in VOF (volume of fluid) or Level Set methods, and thus cannot be used in conjunction with these methods. The combination of existing cavitation models with traditional CFD method can only give the distribution of void fraction, and cannot reflect the dynamic behavior of bubbles during the cavitation. More importantly, it is very difficult for these models to predict thin liquid film between cavitation area and solid wall for the cavitating flow in microchannel. This will lead to an unbelievable wall temperature in the heat transfer problem.

Lattice Boltzmann method (LBM), as a mesoscopic method, has displayed a certain potential in the simulation of fluid flow. It has been proved that Navier-Stokes equation can be derived from the

* Corresponding author at: Institute of Engineering Thermophysics, Chinese Academy of Sciences, Beijing 100190, China.
E-mail address: caijun@mail.etp.ac.cn (J. Cai).

https://doi.org/10.1016/j.ijheatmasstransfer.2018.06.146
0017-9310/

lattice Boltzmann equation [14,15], which indicates that they have the same thermodynamic and physical foundations. The proposal of a multiphase lattice Boltzmann model by Shan and Chen [16] makes it become possible for LBM to simulate two-phase or even multiphase flow. Based on Shan-Chen (SC) single-component multiphase LB model, Sukop et al. [17] simulated the evolution of a cavitation bubble in static liquid, which showed that it was possible for LBM to simulate more complex problems. Subsequently, using two-phase LB model, Mukherjee et al. [18] conducted a simulation of two-phase flow with high density ratio in axially symmetric geometry. Based on this, Chen et al. [19] successfully introduced SC single-component multiphase LB model into the simulation of cavitation bubble growth in rest and shear flows. Nevertheless, it should be noted that an initial seed bubble or nuclei is necessary for above-mentioned LB model. Aiming at this problem, Cheng et al. [20] developed a newly phase-change LB model, which could directly simulate liquid-vapor phase change process by means of a thermodynamic relation determined by the equation of state, instead of initializing a seed bubble. Subsequently, they used this model to successfully simulate the periodic bubble nucleation, growth and departure from a heated surface in pool boiling. And this is a milestone for the development of LBM [21].

Throughout the current studies, LBM has showed a certain potential in the simulation of cavitation bubble. However, the existing researches are still confined to the simulation of single bubble. So far, LBM has not been used to simulate the hydraulic cavitating flow in microchannel. Considering the shortcomings of traditional CFD method in simulating the cavitating flow, this paper employs the phase-change LB model proposed by Cheng et al. [21] to simulate the hydraulic cavitating flow in a microchannel. The validity of the model is checked by both Laplace law and experimental photo of cavitating flow pattern in Ref. [22]. On this basis, we study the effect of fluid-solid interaction strength (g_s) on the simulation of thin liquid film, and find a critical value of g_s to successfully capture thin liquid film. Next, a simple simulation on the contact angel (θ) of droplet on the solid surface is conducted to explain the reason for the existence of the critical value. Overall, the present work lays a good foundation to study the coupling problem of hydraulic cavitating flow and heat transfer using LBM in future.

2. Single-component multiphase LB model

In the present work, hydraulic cavitating flow is induced by the contraction structure rather than by the absorption of heat. Therefore, only one particle distribution function, i.e., the density distribution function, is required, and the temperature distribution function is not concerned.

2.1. Lattice Boltzmann equation

In LBM, flow and heat transfer of fluid are described by the evolution of particle distribution functions. With BGK approximation [23], the evolution equation of the density distribution function is expressed as

$$f_i(\mathbf{x}+\mathbf{e}_i\delta_t, t+\delta_t) - f_i(\mathbf{x},t) = -\frac{1}{\tau}[f_i(\mathbf{x},t) - f_i^{eq}(\mathbf{x},t)] + \Delta f_i(\mathbf{x},t) \quad (1)$$

where $f_i(\mathbf{x},t)$ is the particle distribution function with velocity $\mathbf{e}_i$ at position $\mathbf{x}$ and time t, τ is the relaxation time with the value of 1.0, $f_i^{eq}(\mathbf{x},t)$ is the corresponding equilibrium distribution function in the discrete space, given by

$$f_i^{eq} = \rho\omega_i\left[1 + \frac{\mathbf{e}_i\cdot\mathbf{u}}{c_s^2} + \frac{(\mathbf{e}_i\cdot\mathbf{u})^2}{2c_s^4} - \frac{\mathbf{u}^2}{2c_s^2}\right] \quad (2)$$

where ω_i is the weighting coefficient, ρ is the density and c_s is the lattice sound speed, $\Delta f_i(\mathbf{x},t)$ is the term related to the body force.

In the present work, a 2D simulation is conducted. Thus, D2Q9 discrete velocity model is adopted. The weighting coefficient ω_i is given by $\omega_i = 4/9$ for $i = 0$, $\omega_i = 1/9$ for $i = 1, 2, 3, 4$ and $\omega_i = 1/36$ for $i = 5, 6, 7, 8$. The space discrete velocity vector $\mathbf{e}_i$ is given by

$$\mathbf{e}_i = \begin{cases} (0,0), & i=0 \\ c\{\cos[(i-1)\frac{\pi}{2}],\ \sin[(i-1)\frac{\pi}{2}]\}, & i=1\text{–}5 \\ \sqrt{2}c\{\cos[(2i-1)\frac{\pi}{4}],\ \sin[(2i-1)\frac{\pi}{4}]\}, & i=6\text{–}8 \end{cases} \quad (3)$$

where $c = \delta x/\delta t$ is the lattice speed (δx being the lattice spacing and δ_t being the time spacing). In LBM, δ_x and δ_t are usually set as 1.0. Thus $c = 1$. Note that $c_s{}^2 = c^2/3$ in D2Q9 scheme. The kinematic viscosity is given by

$$\upsilon = c_s^2\left(\tau - \frac{1}{2}\right)\delta_t \quad (4)$$

The density and the velocity in Eq. (2) are obtained by

$$\rho = \sum_{i=1}^{8} f_i, \quad \rho\mathbf{u} = \sum_{i=1}^{8}\mathbf{e}_i f_i \quad (5)$$

2.2. Equation of state

According to Ref. [21], the P-R (Peng–Robinson) equation of state (EOS), which is fit for those phase-change mediums with wide temperature variation range, is used in the present simulation, and given by

$$p = \frac{\rho RT}{1-b\rho} - \frac{a\rho^2\alpha(T)}{1+2b\rho-b^2\rho^2} \quad (6)$$

with

$$\alpha(T) = \left[1 + (0.37464 + 1.54226\omega - 0.26992\omega^2)\left(1-\sqrt{T/T_c}\right)\right]^2 \quad (7)$$

where ω is the acentric factor taking 0.344, and other parameters are as follows [24]: $a = 2/49$, $b = 2/21$ and $R = 1$. The critical properties for P-R EOS can be obtained by setting the first and second derivatives of Eq. (6) with respect to density as zero, given by

$$\frac{dp}{d\rho} = 0, \quad \frac{d^2p}{d\rho^2} = 0 \quad (8)$$

The critical parameters can be calculated as

$$p_c = 0.0596, \quad T_c = 0.0729, \quad \rho_c = 2.6609 \quad (9)$$

2.3. Interaction forces model

To reflecting the effect of body force, the key procedure is how to incorporate various interaction forces into LB model. The current commonly-used methods include velocity shifting method [16], discrete force method [25] and exact difference method [26]. Considering calculation precision and numerical stability, we choose the exact difference method to incorporate the body force term $\Delta f_i(\mathbf{x},t)$ in Eq. (1). The body force term is given by

$$\Delta f_i(\mathbf{x},t) = f_i^{eq}[\rho(\mathbf{x},t), \mathbf{u}+\Delta\mathbf{u}] - f_i^{eq}[\rho(\mathbf{x},t), \mathbf{u}] \quad (10)$$

Where $\Delta\mathbf{u} = \mathbf{F}_{\delta t}/\rho$ is the velocity change under the action of body force within time step δ_t, and $\mathbf{F}$ is the synthesis of all body forces, and given by

$$\mathbf{F} = \mathbf{F}_{\text{int}}(\mathbf{x}) + \mathbf{F}_s(\mathbf{x}) + \mathbf{F}_g(\mathbf{x}) \quad (11)$$