International Journal of Heat and Mass Transfer 115 (2017) 158–168

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Numerical simulation of fluid flow and heat transfer in rotating channels using parallel lattice Boltzmann method

Mehaboob Basha [a], Nor Azwadi Che Sidik [a,b,*], M'hamed Beriache [c]

[a] Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 Johor, Malaysia
[b] Malaysia – Japan International Institute of Technology (MJIIT), University Teknologi Malaysia Kuala Lumpur, Jalan Sultan Yahya Petra (Jalan Semarak), 54100 Kuala Lumpur, Malaysia
[c] Department of Mechanical Engineering, Faculty of Technology, University Hassiba Benbouali, Algeria

ARTICLE INFO

Article history:
Received 26 March 2017
Received in revised form 7 July 2017
Accepted 10 July 2017

Keywords:
Parallel simulation
Lattice Boltzmann Method
Rotating channel
Heat transfer
Rotational buoyancy

ABSTRACT

This paper presents a numerical modeling and simulation of incompressible laminar mixed convection in rotating channels using parallel lattice Boltzmann method. Individual distribution functions with D3Q19 and D3Q6 lattice types were considered to solve fluid flow and heat transfer problems, respectively. The Reynolds number was set to 100, and wall-to-inlet fluid density ratio was set to 0.2. Two rotation modes namely orthogonal and parallel modes were considered with rotation number equal to 0.2. LBM code was written in C language and was parallelized using OpenMP libraries. Domain decomposition method of data parallelism was adopted here, and simulations were conducted on a workstation with dual processors and 64 GB RAM. Predicted velocity and temperature fields were found to agree well with velocity and temperature obtained from Fluent.

1. Introduction

Several engineering fluid flow and heat transfer problems involve moving boundaries and rotating fluids. Conventional numerical methods are well established for simulation of wide variety of fluid flow and heat transfer involving moving boundary, slide and rotating domains [1–5]. From past three decades, Lattice Boltzmann method (LBM) has emerged as new numerical method for simulation of fluid flow and heat transfer problems with stationary boundaries, moving boundaries [6–10], sliding mesh [11] and rotating boundary [12]. In LBM, dynamics of flow evolve through fictitious particles collision and redistribution on a lattice grid with pre-defined lattice velocities. Under a low Mach number assumption, Chapman-Enskog analysis [13] of LB equation associates moments of equilibrium particles to physical (macroscopic) fluid flow variables, such as density, velocity and temperature, in Navier-Stokes equations. Flows with moving boundaries are modeled via source terms at the boundary, while sliding mesh and rotating fluid flows are modeled via sources terms in the LB equations. However, applications of LBM for solving rotating fluid flows and heat transfer are limited. Following is the literature summary related to application of LBM for solving laminar and turbulent fluid flow problems with moving and rotating wall.

Application of LBM for moving boundary fluid flow problem was presented by Lallemand and Luo [14]. Moving boundary was modeled considering simple bounce-back and interpolation concept at flat walls. Kao and Yang investigated curved moving boundary treatments in the LBM [15]. Hybrid method for solving swirl and rotating flow for Quasi 3D problems was proposed by Huang et al. [16]. This model works for low Mach number axis symmetric flow, and finite difference method was used to solve swirl velocity and temperature field. Chen [17] simulated compositional convection in rotating annulus cavity using LBM. Two sets of boundary conditions namely (a) horizontal temperature and vertical solutal gradient, (b) vertical temperature and horizontal solutal gradient were considered. It was concluded that rotation was suppressing the convection effect, especially for the first set of boundary condition. In another study, Chen [8] proposed a simple LBM model for simulation of heat transfer inside a rotating disc-cylinder domain. This model was based on vorticity-stream function and was shown to be more efficient, stable for simulation of high Grashof number flows.

Cai [12] studied fluid flow and heat transfer over a rotating cylinder for various rotation numbers. They proposed a second order accurate method to deal with moving curved boundaries. Secondary flow behavior in a transient rotating periodic channel was presented by Zhang et al. [9]. Their study found formation of

* Corresponding author at: Malaysia – Japan International Institute of Technology (MJIIT), University Teknologi Malaysia Kuala Lumpur, Jalan Sultan Yahya Petra (Jalan Semarak), 54100 Kuala Lumpur, Malaysia.

E-mail address: azwadi@mail.fkm.utm.my (N.A.C. Sidik).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2017.07.044
0017-9310/

Nomenclature

BC	boundary condition
BGK	Bhatnaghar, Gross, Krook
c	lattice velocities
Cs	speed of sound
Cp	specific heat
Dh	hydraulic diameter
DR	density ratio
3D	three dimensional
f_i, g_i	particle distribution function
H	channel height
HPC	high performance computing
L	length of the channel
LBM	lattice Boltzmann method
Ma	mach number
MPI	message passing interface
NS	Naiver-Stokes
p	pressure
Rt	rotation radius
x, y, z	co-ordinates
u, v, w	velocities in x, y, z direction, respectively
W	channel width
w_i	lattice weights

Subscript

d	dimensionless
i	i^{th} direction
id	ID of processor
eq	equilibrium
neq	non-equilibrium
o	reference condition
tb	turbulence

List of symbols

δx	lattice size, m
δt	lattice time, s
ρ_0	mean/reference density, kg/m^3
ω	inverse of relaxation time, 1/s
μ	dynamic viscosity, kg m/s^2
μ_t	turbulent dynamic viscosity, kg m/s^2
υ	kinematic viscosity, m/s^2
υ_t	turbulent kinematic viscosity, m/s^2
τ	relaxation time, s
S	strain rate tensor
Π	stress tensor
Π^{neq}	non-equilibrium stress tensor

multiple secondary flow vortexes, which were regulated by a ratio of pressure gradient and centrifugal force in short periodic computational domain. Flow in a bifurcated channel resembling draft tube of a turbine with rotating inflow boundary condition was studied by Qing-Dong [12]. This study gives details of particle distribution functions at the various boundaries. Result showed that the rotating flow causes mal distribution in the bifurcated channel. Recently, Zhang et al. [10] proposed a method for simulation of fluid flow in a sliding mesh domain within a local reference frame context. Direct numerical simulation of 2D rotating cylinder, 2D blade motion in cross flow and 3D turbulent flow past propeller were conducted for validating this model. The proposed model was able to compute fluxes accurately across the interface of local reference frame.

A large eddy simulation of agitated turbulent flow in a tank driven by Rushton and pitched blade turbines was presented by Jos Derksen [18]. Smagorinsky subgrid-scale model was used in this study, and it was found that flow field was accurate but with high magnitude of turbulent kinetic energy in the vicinity of impeller. All simulations were conducted on Beowulf cluster. In another study, Lu et al. [19] used nonuniform grid with arbitrary computation domain to solve agitated turbulent flow in a tank driven by Rushton turbine. This method is found to reduce 75 % of simulation time when compared with high resolution uniform grid. Eggels [20] conducted direct numerical simulation and large eddy simulation of agitated turbulent flow in tank. This was one of the earliest studies, which support application of LBM for simulation of turbulent flow. Several authors have studied rotation induced turbulence [21,22].

From the literature it can be emphasized that application of LBM for simulation of rotating fluid flow and heat transfer is very sparse. However, LBM simulations in the literature considered fully developed flow region at isothermal conditions. Rotating fluid flow and heat transfer in entrance region has many engineering applications such as turbine blade cooling, and electrical winding cooling. Therefore, the objective of this study was to develop an efficient parallel LBM code for simulation of rotating fluid flow and heat transfer using OpenMP library. The parallel LBM model will consider all body forces acting in a non-isothermal rotating fluid flow problem. Two rotation modes namely orthogonal and parallel rotations were considered, in conjunction with stable fluid flow [23] and thermal [24] boundary conditions. Domain decomposition method of data parallelism was adopted.

2. Methodology

Incompressible LBGK model proposed by He et al. [25] is adopted here. In LBM space is discretized into uniform lattice size of δx and velocity is discretized into finite number of velocities $\vec{c}_i$ to form particle distribution functions $f_i(\vec{r}, t)$. The LBGK evolution equation is as follows.

$$f(\overrightarrow{r} + \delta t\overrightarrow{c_i}, t + \delta t) - f(\overrightarrow{r}, t) = -\Omega_i + FT_i, \Omega_i = \omega(f_i - f_i^{eq}) \quad (1)$$

Ω_i is the BGK [26,27] collision operator which defines particle interaction on lattice sites. FT_i represents force term. Flow dynamics evolve through series of collision and streaming of particle distribution functions. During each time step before collision, particle distribution functions are regularized following the method of Latt et al. [23]. Macroscopic variables of the flow are recovered by the moments of particle distribution functions as $\rho = \sum_i f_i^{eq}$, $\rho_0 u_\alpha = \sum_i c_{i\alpha} f_i^{eq} + \delta t \frac{\overrightarrow{F_i}}{2}$ and $\pi_{\alpha\beta} = \sum_i c_{i\alpha} c_{i\beta} f_i$ and equilibrium distribution function is obtained by expansion of Maxwell-Boltzmann equation to second order and it reads as $f_i^{eq} = w_i\left[\rho + \rho_0\left(\frac{1}{C_s^2}\overrightarrow{c_i}, \overrightarrow{u} + \frac{1}{2C_s^2}Q_i, \overrightarrow{u}\,\overrightarrow{u}\right)\right]$. Nonequilibrium stress tensors are needed during evolution and are calculated as $\Pi_{\alpha\beta}^{neq} = \Pi_{\alpha\beta} - \Pi_{\alpha\beta}^{eq}$ Relation between moments of particle distribution functions and macroscopic fluid flow variables can be established through multi-scale Chapman-Enskog analysis of Eq. (1) in which zeroth order term of particle distribution function is equal to equilibrium distribution function ($f_i^0 = f_i^{eq}$) and first order term of particle distribution function through regularization ($f_i^1 \approx f_i^{neq}$) is related to momentum flux tensor at low Mach number [13]. Velocity, Mach number, pressure and Kinematic viscosity of the