International Journal of Heat and Mass Transfer 113 (2017) 217–228

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Numerical study of three-dimensional natural convection in a cubical cavity at high Rayleigh numbers

Peng Wang [a,b], Yonghao Zhang [b], Zhaoli Guo [a,*]

[a] State Key Laboratory of Coal Combustion, Huazhong University of Science and Technology, Wuhan 430074, PR China
[b] James Weir Fluids Laboratory, Department of Mechanical and Aerospace Engineering, University of Strathclyde, Glasgow G1 1XJ, UK

ARTICLE INFO

Article history:
Received 2 March 2017
Received in revised form 12 May 2017
Accepted 15 May 2017
Available online 27 May 2017

Keywords:
Three–dimensional natural convection
Discrete unified gas–kinetic scheme
Heat transfer
Flow characteristic
Nusselt number–Rayleigh number correlation

ABSTRACT

A systematic numerical study of three-dimensional natural convection of air in a differentially heated cubical cavity with Rayleigh number (Ra) up to 10^{10} is performed by using the recently developed coupled discrete unified gas-kinetic scheme. It is found that temperature and velocity boundary layers are developed adjacent to the isothermal walls, and become thinner as Ra increases, while no apparent boundary layer appears near adiabatic walls. Also, the lateral adiabatic walls apparently suppress the convection in the cavity, however, the effect on overall heat transfer decreases with increasing Ra. Moreover, the detailed data of some specific important characteristic quantities is first presented for the cases of high Ra (up to 10^{10}). An exponential scaling law between the Nusselt number and Ra is also found for Ra from 10^3 to 10^{10} for the first time, which is also consistent with the available numerical and experimental data at several specific values of Ra.

1. Introduction

Natural convection flow (NCF) in a differentially heated cubical cavity is one of the fundamental flow configurations in heat transfer and fluid mechanics studies, and it has many significant applications, including air flow in buildings, cooling of electronic devices, and energy storage systems. In recent years, with the rapid advance of the computer technology, direct numerical simulation (DNS) has become a popular and competitive way to study thermal convection flow problems.

The early numerical studies of NCF were usually restricted to two–dimensional (2D) configuration with relatively low Rayleigh numbers (Ra). The pioneering work of de Vahl Davis et al. [1] provided original benchmark solutions for a square 2D cavity with $10^3 \leqslant Ra \leqslant 10^6$; afterward, more accurate results were presented by Hortmann et al. [2] using the multi-grid method with a much finer mesh. Many others have repeated results with Ra up to 10^8 [2–6].

As actual flow is always three–dimensional (3D), many efforts have also been made on 3D simulations. For example, Mallinson et al. [7] investigated the effects of a certain aspect of a ratio on flow patterns with Ra up to 10^6; Fusegi et al. [8] simulated the NCF in an air-filled cubical cavity for Ra of 10^4 and 10^6, and clarified 3D structures of flow and temperature; Labrosse et al. [9] observed the hysteretic behavior by using a pseudo-spectral solver; the 3D cavity of aspect ratio 4 with periodic lateral walls was studied by Trias et al. [10,11], and significant differences were observed in flow dynamics between 2D and 3D results. They also emphasized that the NCF in a 3D cubical cavity with adiabatic lateral walls had received comparatively less attentions [8,12,9,13,14].

The above mentioned numerical simulations of NCF are performed by the traditional computational fluid dynamics (CFD) methods on the basis of the Navier–Stokes equations (NSEs), which are a set of second–order nonlinear partial differential equations (PDEs). Recently, kinetic methods based on the Boltzmann model equation have become an alternative method to the NSEs with some distinctive features. Different from the NSEs with a nonlinear and nonlocal convection term, the Boltzmann equation is a first-order linear PDE, and the nonlinearity resides locally in its collision term. These features make kinetic methods easy to realize and parallelize with high computational efficiency. Many kinetic methods have been recently utilized to simulate NCF problem, such as the lattice Boltzmann methods (LBM) [15–23] and the gas-kinetic scheme [24].

However, up to date, the study of high Ra (up to 10^{10}) NCF in a differentially heated cubical cavity is limited to 2D configuration, while most of the available investigations on 3D NCF are for low –Ra (up to 10^7) flows. For the NCF with high Ra, it requires much

* Corresponding author.
E-mail address: zlguo@hust.edu.cn (Z. Guo).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2017.05.057
0017-9310/

finer mesh adjacent to wall boundary to capture boundary layer than that in the center of cavity. Therefore, an accurate, stable and mesh flexible method is preferable for numerical study of the 3D NCF.

Recently, starting from the Boltzmann model equation, a discrete unified gas–kinetic scheme (DUGKS) was proposed for both hydrodynamic and rarefied flows [25–27]. As a finite–volume (FV) method, DUGKS can be easily implemented on non-uniform or unstructured meshes to satisfy the local accuracy requirement [28–30]. Particularly, although sharing the common kinetic origin, some distinctive features also exist between DUGKS and LBM. In fact, several comparative studies of the standard LBM and DUGKS have been preformed systematically for laminar flows [28,31], turbulent flows [32,33], and natural convection flows [29] in previous work. Generally, for flows without solid boundaries, for example the decaying turbulent flow, the accuracy of standard LBM is slightly higher than the DUGKS [32], while for flows involving solid boundaries, the DUGKS is even more accurate than the standard LBM [28,33]. Furthermore, owing to the semi-implicitness in the construction of gas distribution function at the cell interfaces, the DUGKS is much more stable and robust than LBM [28,29,32]. However, with a same regular grid, the standard LBM is faster than the DUGKS per iteration [28,29]. But benefiting from the FV nature, non-uniform meshes can be easily employed without loss of accuracy and additional efforts in DUGKS, and its efficiency can be significantly improved by employing a non-uniform mesh according to the local accuracy requirement [28,29,32]. This is the main reason why we use the DUGKS, instead of the LBM, to study the high Rayleigh number natural convection flow, which requires much fine mesh near walls to resolve the thin boundary layers. Although the standard LBM has low numerical dissipation [34], it can only be implemented on regular meshes due to its special streaming process, and the existing reported studies of 3D NCF are limited to low *Ra* [17,23]. Some FV based LBM have been also developed in the past decades, but it has been demonstrated that the DUGKS is obviously superior to the current best FV-LBM [35] in terms of accuracy and numerical stability [31]. In addition, the kinetic nature makes DUGKS suitable for parallelized computing. High computational efficiency is essential for large scale 3D simulations. In order to simulate the incompressible thermal flow, a coupled DUGKS (CDUGKS) has been proposed using the double distribution strategy, and its accuracy, efficiency and numerical stability have been validated by simulating the 2D NCF with *Ra* up to 10^{10} [29]. In this work, we will contribute to study the NCF in a differentially heated cubical cavity with *Ra* up to 10^{10} using the CDUGKS. The method is firstly validated by comparing with available numerical and experimental data. Flow characteristics and heat transfer are then to be investigated. Finally, a scaling correlation between Rayleigh and Nusselt numbers with *Ra* up to 10^{10} will be obtained for the first time.

2. Numerical method

2.1. Kinetic model equations

The coupled discrete unified gas-kinetic scheme is derived from the following Boltzmann model equations [24]

$$\frac{\partial f}{\partial t}+\boldsymbol{\xi}\cdot\nabla_x f=\Omega\equiv\frac{f^{eq}-f}{\tau_v}+F, \tag{1}$$

$$\frac{\partial g}{\partial t}+\boldsymbol{\xi}\cdot\nabla_x g=\Psi\equiv\frac{g^{eq}-g}{\tau_c}, \tag{2}$$

where f and g are gas distribution functions for velocity and temperature fields, respectively, and both are functions of space $\boldsymbol{x}$, time t, and molecular velocity $\boldsymbol{\xi}$; f^{eq} and g^{eq} are the corresponding equilibrium states

$$f^{eq}=\frac{\rho}{(2\pi RT_1)^{D/2}}\exp\left(-\frac{(\boldsymbol{\xi}-\boldsymbol{u})^2}{2RT_1}\right), \tag{3}$$

$$g^{eq}=\frac{T}{(2\pi RT_2)^{D/2}}\exp\left(-\frac{(\boldsymbol{\xi}-\boldsymbol{u})^2}{2RT_2}\right), \tag{4}$$

here R is the gas constant, T_1 and T_2 are the constant variances. For convenience, we set $T_1=T_2$ in this study. $\tau_v=\nu/RT_1$ and $\tau_c=\kappa/RT_2$ are the corresponding relaxation times, here ν and κ are, respectively, the kinematic viscosity and heat conduction coefficient, which determine the Prandtl number $Pr=\nu/\kappa$. For low speed flows, the external force term F can be approximated as [36]

$$F=-\boldsymbol{a}\cdot\nabla_{\xi}f\approx-\boldsymbol{a}\cdot\nabla_{\xi}f^{eq}=\frac{\boldsymbol{a}\cdot(\boldsymbol{\xi}-\boldsymbol{u})}{RT_1}f^{eq}, \tag{5}$$

here $\boldsymbol{a}$ is the acceleration due to buoyancy force, and is approximated by the Boussinesq assumption

$$\boldsymbol{a}=g_0\beta(T-T_0)\hat{\boldsymbol{g}}, \tag{6}$$

where g_0 is the gravitational constant, $\hat{\boldsymbol{g}}$ is the unit vector in the gravitational direction.

In computation, the continuous molecular velocity space should be approximated by a discrete velocity set $\{\xi_i|i\in\mathbb{Z}\}$, so that the integration on molecular velocity space can be numerically computed. For nearly incompressible flow (i.e., when the Mach number $Ma\ll1$), the equilibrium states can be approximated using the Taylor expansion to the second order, i.e.,

$$f_i^{eq}=W_i\rho\left[1+\frac{\boldsymbol{\xi}_i\cdot\boldsymbol{u}}{RT_1}+\frac{(\boldsymbol{\xi}_i\cdot\boldsymbol{u})^2}{2(RT_1)^2}-\frac{|\boldsymbol{u}|^2}{2RT_1}\right], \tag{7}$$

$$g_i^{eq}=W_iT\left[1+\frac{\boldsymbol{\xi}_i\cdot\boldsymbol{u}}{RT_2}+\frac{(\boldsymbol{\xi}_i\cdot\boldsymbol{u})^2}{2(RT_2)^2}-\frac{|\boldsymbol{u}|^2}{2RT_2}\right], \tag{8}$$

where $f_i^{eq}=\omega_i f^{eq}(\xi_i), g_i^{eq}=\omega_i g^{eq}(\xi_i), \omega_i=W_i(2\pi RT_1)^{D/2}\exp\left(\frac{|\xi_i|^2}{2RT_{1,2}}\right)$, and W_i is the weight coefficient corresponding to molecular velocity ξ_i.

In the present study, we use nineteen velocities in three dimensions, i.e., the D3Q19 model, with

$$\boldsymbol{\xi}_i=\begin{cases}(0,0) & i=0\\ (\pm1,0,0)c,(0,\pm1,0)c,(0,0,\pm1)c & i=1-6,\\ (\pm1,\pm1,0)c,(\pm1,0,\pm1)c,(\pm1,\pm1,)c & i=7-18,\end{cases} \tag{9}$$

where $c=\sqrt{3RT_1}$, and the corresponding weight coefficients are $W_0=1/3, W_{1,\ldots,6}=1/18$ and $W_{7,\ldots,18}=1/36$. The discrete distribution functions $f_i(\boldsymbol{x},t)=\omega_i f(\boldsymbol{x},\boldsymbol{\xi}_i,t)$ and $g_i(\boldsymbol{x},t)=\omega_i g(\boldsymbol{x},\boldsymbol{\xi}_i,t)$ satisfy the following equations

$$\frac{\partial f_i}{\partial t}+\boldsymbol{\xi}_i\cdot\nabla_x f_i=\Omega_i\equiv\frac{f_i^{eq}-f_i}{\tau_v}+F_i, \tag{10}$$

$$\frac{\partial g_i}{\partial t}+\boldsymbol{\xi}_i\cdot\nabla_x g_i=\Psi_i\equiv\frac{g_i^{eq}-g_i}{\tau_c}. \tag{11}$$

The fluid density, velocity, and temperature can be obtained from the discrete distribution functions,

$$\rho=\sum_i f_i,\quad \rho\boldsymbol{u}=\sum_i\boldsymbol{\xi}_i f_i,\quad T=\sum_i g_i. \tag{12}$$

2.2. DUGKS for velocity field

The DUGKS is a FV method in which the computational domain is divided into a set of control volumes. We integrate Eq. (10) over