
This version is available at https://strathprints.strath.ac.uk/60895/

Strathprints is designed to allow users to access the research output of the University of Strathclyde. Unless otherwise explicitly stated on the manuscript, Copyright © and Moral Rights for the papers on this site are retained by the individual authors and/or other copyright owners. Please check the manuscript for details of any other licences that may have been applied. You may not engage in further distribution of the material for any profitmaking activities or any commercial gain. You may freely distribute both the url (https://strathprints.strath.ac.uk/) and the content of this paper for research or private study, educational, or not-for-profit purposes without prior permission or charge.

Any correspondence concerning this service should be sent to the Strathprints administrator: strathprints@strath.ac.uk

The Strathprints institutional repository (https://strathprints.strath.ac.uk) is a digital archive of University of Strathclyde research outputs. It has been developed to disseminate open access research outputs, expose data about those outputs, and enable the management and persistent access to Strathclyde's intellectual output.
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

A R T I C L E   I N F O

Article info:
Received 2 March 2017
Received in revised form 12 May 2017
Accepted 15 May 2017

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

A B S T R A C T

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 $10^{4}$ to $10^{10}$ for the first time, which is also consistent with the available numerical and experimental data at several specific values of $Ra$.

© 2017 Elsevier Ltd. All rights reserved.

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 $10^{3} \leq Ra \leq 10^{5}$; 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 $10^{8}$; 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 attention [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).
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 performed 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 LB 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 LB, to study the high speed flows, the external force term F can be approximated by a discrete velocity set [28–30], it can only be slightly higher than the DUGKS but benefiting from the FV nature, the standard LB is faster than the DUGKS.

The coupled discrete unified gas-kinetic scheme is derived from the following Boltzmann model equations [24]
\[
\frac{\partial f}{\partial t} + \xi \cdot \nabla f = \Omega \equiv \frac{f^q - f}{\tau_0} + F, 
\]
\[
\frac{\partial g}{\partial t} + \xi \cdot \nabla g = \Psi \equiv \frac{g^q - g}{\tau_c},
\]
where f and g are gas distribution functions for velocity and temperature fields, respectively, and both are functions of space x, time t, and molecular velocity $\xi_j$ and $g^q$ are the corresponding equilibrium states
\[
f^q = \frac{\rho}{(2\pi RT_1)^{3/2}} \exp \left(-\frac{(\xi - \bar{u})^2}{2RT_1}\right), \quad (3)
\]
\[
g^q = \frac{T}{(2\pi RT_2)^{3/2}} \exp \left(-\frac{(\xi - \bar{u})^2}{2RT_2}\right), \quad (4)
\]
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. $\tau_c = \kappa / RT_1$ and $\tau_0$ are the corresponding relaxation times, here $\nu$ and $\kappa$ 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]
\[
F = -a \cdot \nabla f \approx -a \cdot \nabla f^{eq} = \frac{a \cdot (\xi - \bar{u})}{RT_1} f^{eq},
\]
here a is the acceleration due to buoyancy force, and is approximated by the Boussinesq assumption
\[
a = g_0(|T - T_0|),
\]
where $g_0$ is the gravitational constant, $\bar{g}$ is the unit vector in the gravitational direction.

In computation, the continuous molecular velocity space can be approximated by a discrete velocity set ($\xi_j \in \mathbb{Z}$), so that the integration on molecular velocity space can be numerically computed. For nearly incompressible flow (i.e., when the Mach number $Ma < 1$), the equilibrium states can be approximated using the Taylor expansion to the second order, i.e.,
\[
f^{eq}_i = W_i \rho \left[1 + \frac{\xi_j - \bar{u}}{RT_1} + \frac{(\xi_j - \bar{u})^2}{2(\nu RT_1)^2} \frac{\mu^2}{RT_1}\right], \quad (7)
\]
\[
g^{eq}_i = W_i T \left[1 + \frac{\xi_j - \bar{u}}{RT_2} + \frac{(\xi_j - \bar{u})^2}{2(\kappa RT_2)^2} \frac{\mu^2}{RT_2}\right], \quad (8)
\]
where $f^{eq}_i = \omega_i f^{eq}(\xi_i), g^{eq}_i = \omega_i g^{eq}(\xi_i), \omega_i = W_i(2\pi RT_1)^{3/2} \exp \left(\frac{|\xi^2|}{2RT_1}\right)$, and $W_i$ is the weight coefficient corresponding to molecular velocity $\xi_i$.

In the present study, we use nineteen velocities in three dimensions, i.e., the D3Q19 model, with
\[
\xi_i = \begin{cases} 
(0,0,0) & i = 0 \\
(\pm 1, 0, 0), (0, \pm 1, 0), (0, 0, \pm 1) & i = 1 - 6 \\
(\pm 1, \pm 1, 0), (\pm 1, 0, \pm 1), (0, \pm 1, \pm 1) & i = 7 - 18,
\end{cases}
\]
where $c = \sqrt{3RT_1}$, and the corresponding weight coefficients are $W_0 = 1/3, W_{1-6} = 1/18$ and $W_{7-18} = 1/36$. The discrete distribution functions $f_i(x, t) = \omega_i f_i(x, \xi, t)$ and $g_i(x, t) = \omega_i g_i(x, \xi, t)$ satisfy the following equations
\[
\frac{\partial f_i}{\partial t} + \xi_i \cdot \nabla f_i = \Omega_i \equiv \frac{f^{eq}_i - f_i}{\tau_0} + F_i, \quad (10)
\]
\[
\frac{\partial g_i}{\partial t} + \xi_i \cdot \nabla g_i = \Psi_i \equiv \frac{g^{eq}_i - g_i}{\tau_c}. \quad (11)
\]

The fluid density, velocity, and temperature can be obtained from the discrete distribution functions,
\[
\rho = \sum_i f_i, \quad \rho u = \sum_i \xi_i f_i, \quad T = \sum_i g_i. \quad (12)
\]

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
a control volume $V_i$ centered at $x_i$ from $t_n$ to $t_{n+1}$ (the time step $\Delta t = t_{n+1} - t_n$ is assumed to be a constant), and use the midpoint and trapezoidal rules for time integrations of the convection and collision terms, respectively, so we get the updating equation as [25]

$$f_{i}^{n+1} = f_{i}^{n} - \frac{\Delta t}{V_i} F_{i}^{n+1/2},$$

(13)

where

$$F_{i}^{n+1/2} = \int_{\partial V_i} (\xi \cdot \mathbf{n}) f_{i}(x, t_{n+1/2}) dS,$$

(14)

is the micro–flux across the cell interface, and

$$\tilde{f}_i = f_i - \frac{\Delta t}{2} \Omega_i, \quad \tilde{f}_i' = f_i + \frac{\Delta t}{2} \Omega_i.$$  \hspace{1cm} (15)

Since the collision term $\Omega_i$ is mass and momentum conservative, the density $\rho$ and velocity $\mathbf{u}$ can be computed by

$$\rho = \sum_i \tilde{f}_i, \quad \rho \mathbf{u} = \sum_i \tilde{f}_i \Omega_i = \sum_i \tilde{f}_i' \Omega_i.$$  \hspace{1cm} (16)

The key ingredient in updating $f_i$ is to evaluate the interface flux $F_{i}^{n+1/2}$, which is solely determined by the distribution function $f_i(x, t_{n+1/2})$. In DUGKS, after integrating Eq. (10) along the characteristic line of molecule within a half time step ($h = \Delta t/2$), the evaluation of $f_i(x, t_{n+1/2})$ can be traced back to the interior of neighboring cells,

$$f_i(x_i, t_n + h) = f_i(x_i, t_n) - h \xi_i \cdot \mathbf{u}_b,$$

(17)

where

$$f_i = f_i - \frac{\Delta t}{2} \Omega_i, \quad f_i' = f_i + \frac{\Delta t}{2} \Omega_i,$$

(18)

and the gradient $\mathbf{u}_b = \nabla f_i(x_i, t_n)$ can be approximated by linear interpolation.

Again using the conservative property of $\Omega_i$, the density $\rho$ and velocity $\mathbf{u}$ at the cell interface can be obtained from

$$\rho = \sum_i \tilde{f}_i, \quad \rho \mathbf{u} = \sum_i \tilde{f}_i \Omega_i = \sum_i \tilde{f}_i' \Omega_i,$$

(19)

from which the equilibrium distribution function $f_i^{eq}(x_i, t^n + h)$ is determined. Therefore, based on Eq. (18) and the obtained equilibrium state, the original distribution function $f_i$ at the cell interface can be determined from $f_i'$,

$$f_i(x_i, t_n + h) = \frac{2\tau_e}{2\tau_e + h} f_i'(x_i, t_n + h) + \frac{h}{2\tau_e + h} f_i^{eq}(x_i, t_n + h)$$

$$+ \frac{\tau_e h}{2\tau_e + h} f_i,$$

(20)

from which the micro–flux Eq. (14) can be evaluated.

In computation, we only need to follow the evolution of $\tilde{f}_i$ in Eq. (13). The required variables for its evolution are determined by

$$\tilde{f}_i^+ = \frac{2\tau_e}{2\tau_e + h} \tilde{f}_i + \frac{3h}{2\tau_e + h} f_i^{eq} + \frac{3\tau_e h}{2\tau_e + h} F,$$

(21)

$$\tilde{f}_i^- = \frac{4}{3} \tilde{f}_i^+ - \frac{1}{3} \tilde{f}_i.$$  \hspace{1cm} (22)

2.3. DUGKS for temperature field

DUGKS for Eq. (11) can be constructed similarly as for Eq. (10). Eq. (11) is firstly integrated in the same control volume $V_i$ from $t_n$ to $t_{n+1}$, then, the same integration rules are employed to approximate the time integrations of convection and collision terms, one can get the updating equation for $g_r$,

$$\hat{g}_{i,k}^{n+1} = \hat{g}_{i,k}^n - \frac{\Delta t}{|V_i|} F_{i,k}^{n+1/2},$$

(23)

where $F$ is the micro–flux,

$$F_{i,k}^{n+1/2} = \int_{\partial V_i} (\xi \cdot \mathbf{n}) g_{i,k}(x, t_{n+1/2}) dS,$$

(24)

and

$$\hat{g}_i = g_i - \frac{\Delta t}{2} \Psi_i, \quad \hat{g}_i' = g_i + \frac{\Delta t}{2} \Psi_i.$$  \hspace{1cm} (25)

are auxiliary distribution functions. Based on the conservative property of $\Psi_i$, the temperature can be computed as

$$T = \sum_i \hat{g}_i.$$  \hspace{1cm} (26)

In order to evaluate the micro–flux $F$, we again integrate Eq. (11) within a half time step $h$ along the characteristic line with an end point $x_i$ at the cell interface, and use the trapezoidal rule to approximate the time integration of collision term,

$$\hat{g}_i(x_i, t_n + h) = \hat{g}_i^+(x_i - h \xi_i, t_n) = \hat{g}_i^+ (x_i, t_n) - h \xi_i \cdot \mathbf{u}_b,$$

(27)

where

$$\hat{g}_i = g_i - \frac{h}{2} \Psi_i, \quad \hat{g}_i' = g_i + \frac{h}{2} \Psi_i,$$

(28)

and $\mathbf{u}_b = \nabla \hat{g}_i(x_i, t_n)$. The Taylor expansion for $\hat{g}_i^+(x_i - h \xi_i, t_n)$ is made around the cell interface $x_i$. Based on Eq. (28) and the compatibility condition, the temperature at the cell interface can be computed as

$$T = \sum_i \hat{g}_i.$$  \hspace{1cm} (29)

Together with the conserved variables in velocity field, the equilibrium distribution function $g_i^{eq}(x_i, t^n + h)$ can be fully determined. Then, the original distribution function $g_i$ can be obtained,

$$g_i(x_i, t_n + h) = \frac{2\tau_e}{2\tau_e + h} \hat{g}_i(x_i, t_n + h) + \frac{h}{2\tau_e + h} g_i^{eq}(x_i, t_n + h),$$

(30)

from which the interface numerical flux $F$ is determined.

In computation, it only needs to follow the evolution of $\hat{g}$ according to Eq. (23). The required variables for its evolution are determined by

$$\hat{g}_i^+ = \frac{2\tau_e}{2\tau_e + h} \hat{g}_i + \frac{3h}{2\tau_e + h} g_i^{eq},$$

(31)

$$\hat{g}_i^- = 4 \hat{g}_i^+ - \frac{1}{3} \hat{g}_i.$$  \hspace{1cm} (32)

2.4. Kinetic boundary conditions

In this study, kinetic boundary conditions for velocity and temperature fields are required. For the velocity field, no–slip boundary on the fixed wall ($x_w$) can be realized by the bounce–back rule, which gives

$$f_i(x_w, t + h) = f_i(x_w, t + h), \quad \xi \cdot \mathbf{n} > 0,$$

(33)

where $\mathbf{n}$ is the unit vector normal to the wall pointing to the cell.

As for the temperature field, two types of boundaries i.e., fixed temperature and adiabatic walls are considered. For the constant temperature boundary, the distribution function for molecule leaving the wall can be given by [37]
2.3. Algorithm

For clarity, we now list the computational procedure of DUGKS. Specifically, with initialized \( f_i^n \) and \( g_{ij}^n \) in all cells centered at \( \mathbf{x}_i \), the procedure of the DUGKS at each time \( t_n \) reads as follows:

1. Compute the distribution functions \( f_{ij}^{n+1/2} (\mathbf{x}) \) (Eq. (21)) and \( g_{ij}^{n+1} (\mathbf{x}) \) (Eq. (31)) in each cell.
2. Compute the distribution functions \( f_i^{n+1/2} (\mathbf{x}_i) \) (Eq. (17)) and \( g_{i}^{n+1/2} (\mathbf{x}_i) \) (Eq. (27)).
3. Compute the original distribution functions \( f_i^{n+1/2} (\mathbf{x}_i) \) (Eq. (20)) and \( f_i^{n+1/2} (\mathbf{x}_i) \) (Eq. (30)).
4. Compute the microflux across the cell interfaces from \( f_{ij}^{n+1/2} (\mathbf{x}_i) \) (Eq. (14)) and \( f_i^{n+1/2} (\mathbf{x}_i) \) (Eq. (24)).
5. Update the distribution functions \( f_{ij}^{n+1} \) and \( g_{ij}^{n+1} \) via Eqs. (13) and (23), respectively, where \( f^+ \) and \( g^+ \) are computed respectively according to Eqs. (22) and (32).

3. Numerical results

In this work, natural convection of air in a differentially heated cubical cavity with the adiabatic lateral walls is studied in 3D. As illustrated in Fig. 1, the configuration is a cubical box with a cold wall \( (T = T_h) \) on the left side \( (x = 0) \) and a hot wall \( (T = T_c) \) on the right side \( (x = H) \), and the other four walls are adiabatic. The gravity is along the \( y \)-direction. The Rayleigh number is defined as \( Ra = \frac{g \beta \Delta T H^3}{\nu \kappa} \), where \( \Delta T = T_h - T_c \) is the temperature difference between the hot and cold walls, and \( H \) is the length of the cavity.

In the simulations, we set the length of the box \( H = 1 \), the Prandtl number \( Pr = 0.71 \), the temperature \( T_c = 0 \) and \( T_h = 1 \) on the cold and hot walls, respectively. In addition, we set \( RT_1 = 10 \) and \( g_0 \beta = 0.1 \) so that the Mach number \( Ma = u_0 / \sqrt{RT_1} \) is small enough to satisfy the incompressible approximation, where \( u_0 = \sqrt{g \beta \Delta T H} \) is the characteristic velocity. The velocities are normalized by the characteristic velocity \( u_0 \) unless otherwise stated. The velocity boundary condition, Eq. (33), is applied to all walls; the temperature boundary conditions, Eqs. (34) and (35), are applied to the isothermal and adiabatic walls, respectively. The time step is determined by the Courant–Friedrichs–Lewy (CFL) condition, i.e., \( \Delta t = \eta \Delta x_{min} / \zeta_{max} \), where \( \eta \) is the CFL number, \( \Delta x_{min} \) is minimum grid spacing, and \( \zeta_{max} \) is the maximum discrete molecular velocity. We set the CFL number \( \eta = 0.95 \) in the following simulations.

The FV nature of DUGKS make it easy to vary mesh resolution according to the local accuracy requirement. In our simulations, a set of non-uniform meshes are adopted with \( N \) grid points in each direction, and the mesh gradually becomes finer from the center to walls of the cavity. The location of a control volume center...
\( (x, y, z) \) is generated by 
\( x_i = (\zeta_i + \zeta_{i+1})/2, \quad y_j = (\zeta_j + \zeta_{j+1})/2, \quad z_k = (\zeta_k + \zeta_{k+1})/2, 0 \leq i, j, k < N, \) 
where \( \zeta_i \) is defined by
\[ \zeta_i = 1 + \frac{\tanh(a(i/N - 0.5))}{\tanh(a/2)}, \quad i = 0, 1, 2, \ldots, N - 1, \] (36)
in which \( a \) is a constant that determines the mesh distribution. Here \( a \) is set to 2.5, and the mesh distribution is shown in Fig. 2. In our simulations, a suitable mesh should be adopted to make a balance between computational accuracy and efficiency. For the cases of 
\( 10^3 \leq Ra \leq 10^8, Ra = 10^7, \) and 
\( 10^8 \leq Ra \leq 10^{10}, \) we use 50x100x3 and 200x3 mesh points, respectively. The grid independence tests show that the given mesh resolutions are adequate. All simulations are performed by using the Message Passing Interface (MPI) technique on a cluster (Dual Intel Xeon CPU E5-2680 v3 with 64 GB RAM memory).

In order to make a quantitative study, we will measure some important quantities of the NCF, including the maximum horizontal velocity at the vertical centerline of the midplane \( z = H/2, \) \( u_{\text{max}} \), and the corresponding \( y \)-coordinate, the maximum vertical velocity at the horizontal centerline of the midplane \( x = H/2, \) \( v_{\text{max}} \), and the corresponding \( x \)-coordinate, the mean Nusselt num-

![Fig. 3. Temperature distribution on the symmetry plane of \( z = 0.5 \) for \( Ra = 10^7 \). The finite–difference (3D FD) result [8] and the experimental data (Exp.) [38] at 1.89 x 10^5 are included for comparison.](image)

![Fig. 4. Velocity distribution on the symmetry plane of \( z = 0.5 \) for \( Ra = 10^7 \). The finite–difference result (3D FD) [8] and experimental data (Exp.a [39]: \( Ra = 1.03 \times 10^7 \), Exp.b [38]: \( Ra = 1.89 \times 10^7 \)) are included.](image)
ber, $Nu_m$ and the overall Nusselt number, $Nu_o$, where $Nu_m$ and $Nu_o$ are defined by

$$Nu_m(z) = \int_{0}^{H} Nu(l,y,z)|_{y=0 \text{ or } x=0} dy,$$

$$Nu_l(y,z) = \frac{\partial T(y,z)}{\partial x}.$$  \(37\)

$$Nu_o = \int_{0}^{H} Nu_m(z)dz,$$  \(38\)

in which $Nu_l$ is the local Nusselt number.

### 3.1. Validation

The comparative study with the available results in the literature is firstly performed to validate the 3D CDUGKS code. Initially, we set the flow velocity $u = 0$ and the temperature $T = 0.5$ for the whole flow field. To assess whether the flows have reached the steady-state, the following criterion is applied,

$$\sqrt{\sum ||u(t) - u(t - 1000\Delta t)||^2} < 10^{-8}. \quad \text{\(39\)}$$

Table 1 shows the quantities of interest on the symmetry plane of $z = 0.5$, as well as the mean $Nu$ ($Nu_m$) and the overall $Nu$ ($Nu_o$) on the hot wall for $10^3 \leq Ra \leq 10^6$. The results of a high-resolution finite-difference (FD) method [8] are included for comparison. As can be seen, good agreements are achieved with the benchmark results, and the maximum relative error is less than 5%, which is acceptable for engineering applications. Also, it is found that similar to the LBM [17], the CDUGKS almost overall underestimates the results of these quantities when compared with the same reference solution. The deviation can be attributed to the difference in the numerical methods. The reference solutions were obtained by a third-order QUICK scheme for the incompressible Navier-Stokes equations on a fine non-uniform mesh of $62^3$, while the DUGKS is a second-order accurate method, and a relatively coarse non-uniform mesh of $50^3$ is employed in the validation. However, the deviations between the results of DUGKS and reference solutions are still acceptable (less than 5%).

In addition, we compare the temperature and velocity profiles given by CDUGKS with those from the experiments or the NS solver that are available at some specific values of $Ra$. It should be noted that cavities with large depth aspect ratios were usually employed in the experiments, which precludes the precise comparisons of each set of data. Fig. 3 shows the temperature profiles on the symmetry plane $z = 0.5$ for the case of $Ra = 10^5$, in which the experimental data at $Ra = 1.89 \times 10^5$ acquired by a Mach-Zehnder interferometry technique [38] and the NS solver results obtained by FD method [8] are also included for comparison. It is found that

![Fig. 5. Isotherms on the symmetry planes for $10^3 \leq Ra \leq 10^6$.](image-url)
the respective temperature distributions at the mid–lines $y = 0.5$ and $x = 0.5$ agree well with the FD results [8], and are in good agreement with the experimental data at the mid-line $y = 0.5$. However, there is a noticeable discrepancy at the mid–line $x = 0.5$ between the two numerical and experimental results, and the derivation increases when approaching the horizontal walls. This may be attributed to the unavoidable heat transfer through adiabatic walls in the experimental situations [38] and slight difference in $Ra$ between the numerical simulations and experiment.

Fig. 4 shows the velocity profiles on the symmetry plane $z = 0.5$ for the case of $Ra = 10^7$. In addition to the above mentioned reference results, another set of experimental data at $Ra = 1.03 \times 10^5$ [39] are also included. It is clearly seen that the CDUGKS results are in excellent agreement with the FD results, and agree reasonably well with two sets of experimental data, despite slight deviations in the peak values of velocities.

3.2. Flow characteristics and heat transfer for $10^3 \leq Ra \leq 10^{10}$

The flow characteristics and heat transfer are two important features of the 3D NCF, but systematic investigation is still lack in the literature, especially for the cases of high $Ra$ (larger than $10^6$). In this subsection, a detailed study of the flow characteristics and heat transfer of NCF in a 3D cavity is performed for $10^3 \leq Ra \leq 10^{10}$.

Figs. 5 and 6 show isotherms or instantaneous isotherms on the symmetry planes for $10^3 \leq Ra \leq 10^6$ and $10^7 \leq Ra \leq 10^{10}$, respectively. It can be seen that as $Ra$ increases, isotherms change from almost vertical to be horizontal in the center of the symmetry plane $z = 0.5$, and are vertical only adjacent to the isothermal walls, which indicates that the dominant heat transfer mechanism changes from conduction to convection. In addition, temperature boundary layers are developed near the isothermal walls, and become thinner as $Ra$ increases. However, such phenomenon does not appear near the adiabatic walls. Furthermore, as $Ra$ reaches to $10^7$ and $10^{10}$, the isotherms become oscillatory, which means that the heat transfer has turned to be time–dependent. Moreover, as shown on the symmetry planes of $y = 0.5$ and $x = 0.5$, the temperature adjacent to the lateral adiabatic walls is lower than that in the center region, suggesting that the adiabatic walls inhibit heat transfer, but such effects reduce as $Ra$ increases. It is also noted that for the cases of $Ra \leq 10^6$, isotherms on the symmetry plane of $z = 0.5$ are very similar to the 2D results [17,40].

Figs. 7 and 8 show velocity magnitude distributions on the symmetry planes for $10^3 \leq Ra \leq 10^6$ and $10^7 \leq Ra \leq 10^{10}$, respectively. We observe that the flow motion in the 3D cavity is complicated due to the effect of lateral walls. For the cases of low $Ra$, flow run-
nels are developed near the center of walls, and as $Ra$ increases, they move to the corners of the cavity. For example, as shown in Fig. 8(a), the maximum velocity is located adjacent to two corners of the isothermal walls, which means two tunnels have been developed there. In addition, we also find that similar to the temperature field, velocity boundary layers are formed near the isothermal walls, which become thinner with increasing $Ra$, but no apparent velocity boundary layers are developed adjacent to the adiabatic walls. Furthermore, when the $Ra$ reaches $10^9$, the flow field is evolving to chaos.

Heat transfer in the cubical cavity is characterized by the Nusselt number ($Nu$), which is a measure of the relative strengths of the convective and conductive heat transfer processes. The heat transfer is dominated by conduction in the flow domain where $Nu < 1$, and by convection as $Nu > 1$. Fig. 9 shows the contours of the local Nusselt number ($Nu_l$) on the hot wall. It can be seen that the value of $Nu$ at the symmetry plane of $z = 0.5$ is higher than the other regions, suggesting that the lateral adiabatic walls suppress the convective heat transfer in the 3D cavity, but such effect is gradually reduced as $Ra$ increases. It is also observed that the higher local Nusselt number appears near the bottom of hot wall, which means that convection is the dominant heat transfer process in this region. This is because that with anticlockwise vortex rolling, the cold fluid near the bottom of cavity is closer to the hot wall than that in the vicinity of upper region, see Figs. 5 and 6, so the convective heat transfer is more intensive at the bottom of hot wall.

Quantitatively, we present time–averaged characteristic quantities of interest on the symmetry plane of $z = 0.5$ for $10^7 \leq Ra \leq 10^{10}$ in Table 2, which is absent from the literatures. Note that the provided results are already mesh independent, and the time–averaged operation is preformed over a long enough period. It is found that the maximum horizontal velocity is located near the bottom adiabatic wall, and the maximum vertical velocity moves to the hot wall as $Ra$ increases. Also, the vertical velocity near the isothermal hot wall is about one order of magnitude larger than the horizontal one near the adiabatic wall, which indicates that contrary to the results near the adiabatic wall, a boundary layer is formed adjacent to the isothermal wall. This result is consistent with the phenomenon we have observed in Fig. 7. Moreover, the conservation of heat transfer in the cubical cavity is critical to the DNS of high $Ra$ NCF. Therefore, the mean and overall Nusselt numbers on cold and hot walls are included in Table 2 to validate the conservation property of our simulation. It is found that the maximum difference of $Nu_m$ or $Nu_0$ between the cold and hot walls is less than 0.5%, suggesting that the heat transfer conservation in the cavity is well predicted by the CDUGKS.

3.3. $Nu$–$Ra$ correlation

$Ra$ and $Nu$ are two important parameters in description of heat transfer. For the 3D NCF in a cubical cavity, it has been demonstrated that there is an exponential relation between $Nu$ and $Ra$. However, no such result is available from the literatures. For the 3D NCF in a cubical cavity, the following relation is obtained:

$$Nu = 0.13_0.15 \cdot 0.02 \cdot 0.01 \cdot 0.03 \cdot 0.04 \cdot 0.05 \cdot 0.08 \cdot 0.12 \cdot 0.14 \cdot 0.16 \cdot 0.2 \cdot 0.22 \cdot 0.24 \cdot 0.26$$

$Ra = 10^3$ to $Ra = 10^6$. 

Fig. 7. Velocity magnitude contours on the symmetry planes for $10^3 \leq Ra \leq 10^6$. 

$$\begin{array}{ll}
(a) Ra=10^3 & \hspace{1cm} (b) Ra=10^4 \\
(c) Ra=10^5 & \hspace{1cm} (d) Ra=10^6
\end{array}$$
for $10^3 < Ra < 10^7$ [8], but it is yet to be investigated for the higher $Ra$. Based on the present simulation results, the scaling laws to correlate $Ra$ and $Nu$ with $Ra$ up to $10^{10}$ can be obtained as below

$$
\begin{align*}
Nu_m &= 0.1522Ra^{0.2942} & & 10^3 < Ra < 10^7, \\
Nu_m &= 0.3533Ra^{0.2395} & & 10^7 < Ra < 10^{10}.
\end{align*}
$$

and

$$
\begin{align*}
Nu_o &= 0.1270Ra^{0.3052} & & 10^3 < Ra < 10^7, \\
Nu_o &= 0.3408Ra^{0.2410} & & 10^7 < Ra < 10^{10}.
\end{align*}
$$

In Fig. 10, the fitting relationships of $Nu_m - Ra$ (Eq. (40)) and $Nu_o - Ra$ (Eq. (41)), as well as the numerical results computed by CDUGKS, are shown. The available numerical and experimental data at some specific values of $Ra$ are also included. As shown, the predicted results given by Eqs. (40) and (41) are in good agreement with the solutions obtained by the spectral Chebyshev method [13,14] and the experiment data [14], which validates the proposed scaling laws. Note that the scaling relationship for $10^7 < Ra < 10^{10}$ reported here is given for the first time, which may help in thermal engineering applications.

4. Conclusions

In this paper, the natural convection of air in a differentially heated cubical cavity with the adiabatic lateral walls is studied by the coupled discrete unified gas–kinetic scheme (CDUGKS). The validation of our 3D code is carefully performed. It is shown that the CDUGKS results are in good agreement with those from the traditional CFD solvers and experiments. Particularly, the CDUGKS can accurately capture the temperature and velocity boundary layers adjacent to the isothermal walls. In addition, some important quantities computed by the CDUGKS also agree quantitatively well with the available numerical and experimental data.

The flow characteristics are studied systematically for $10^3 < Ra < 10^{10}$. It is found that the temperature and velocity boundary layers are formed near the isothermal walls, and become thinner as $Ra$ increases, while no apparent boundary layer is developed near the adiabatic walls. In addition, it is observed that flow tunnels are developed close to the center of isothermal walls, and they move to the cavity corners in the joint of isothermal and adiabatic walls as $Ra$ increases, meanwhile, the flow evolves from the steady–state to time–dependent state. When $Ra$ approaches to $10^9$, the flow motions turn to be fully turbulent. Moreover, it is also

![Fig. 8. Instantaneous velocity magnitude contours on the symmetry planes for $10^7 < Ra < 10^{10}$](image-url)
found that the lateral adiabatic walls have an inhibition on the distribution of temperature in the cavity.

By studying the local Nusselt number distribution on the hot wall, we observe that the lateral adiabatic walls apparently suppress the heat transfer. Particularly, the Nusselt number on the hot wall has an instant drop near the adiabatic walls, which means that the convective intensity in the center of cavity is stronger than that close to the adiabatic walls. But, the effect of adiabatic walls on overall heat transfer decreases as $Ra$ increases.

Moreover, the time-averaged characteristic quantities of interest on the symmetry plane for the Rayleigh number up to $10^{10}$ is presented for the first time. The results of Nusselt number on the cold and hot walls show that the heat transfer predicted by our simulations is well conserved in the cavity.

Finally, the scaling laws between the Nusselt number (local and overall) and Rayleigh number in the air filled cubical cavity for $Ra$ up to $10^{10}$ are revealed. The predicted results agree well with the available numerical and experimental data at several specific

**Table 2**

<table>
<thead>
<tr>
<th>$Ra$</th>
<th>$10^7$</th>
<th>$10^8$</th>
<th>$10^9$</th>
<th>$10^{10}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u_{max}$</td>
<td>0.0558</td>
<td>0.0443</td>
<td>0.0237</td>
<td>0.0166</td>
</tr>
<tr>
<td>$y$</td>
<td>0.1169</td>
<td>0.0515</td>
<td>0.0513</td>
<td>0.0515</td>
</tr>
<tr>
<td>$v_{max}$</td>
<td>0.2590</td>
<td>0.2619</td>
<td>0.2565</td>
<td>0.2477</td>
</tr>
<tr>
<td>$x$</td>
<td>0.9767</td>
<td>0.9875</td>
<td>0.9938</td>
<td>0.9961</td>
</tr>
<tr>
<td>$Nu_{mc}$</td>
<td>16.3909</td>
<td>29.7261</td>
<td>52.9029</td>
<td>89.0555</td>
</tr>
<tr>
<td>$Nu_{mh}$</td>
<td>16.4153</td>
<td>29.9169</td>
<td>52.3006</td>
<td>89.4952</td>
</tr>
<tr>
<td>$Nu_{oc}$</td>
<td>16.1872</td>
<td>29.6782</td>
<td>52.0079</td>
<td>89.0350</td>
</tr>
<tr>
<td>$Nu_{oh}$</td>
<td>16.2112</td>
<td>29.7141</td>
<td>52.0776</td>
<td>89.2018</td>
</tr>
</tbody>
</table>

**Fig. 9.** Contours of the local Nusselt number ($Nu_l$) on the hot wall: (a) $Ra = 10^4$; (b) $Ra = 10^6$; (c) $Ra = 10^8$; (d) $Ra = 10^{10}$. 
values of $Ra$, which demonstrates the reliability of the new relationships.

**Acknowledgment**

This work is financially supported by the National Key Research Project (2016YFB0600805), and the UKs Engineering and Physical Sciences Research Council (EPSRC) under grants EP/M021475/1, EP/L00030X/1.

**References**


[32] P. Wang, L.-P. Wang, Z. Guo, Comparison of the lattice Boltzmann equation and
discrete unified gas-kinetic scheme methods for direct numerical simulation

[33] Y. Bo, P. Wang, Z. Guo, L.-P. Wang, DUGKS simulations of three-dimensional
Taylor–Green vortex flow and turbulent channel flow, Comput. Fluids
<http://doi.org/10.1016/j.compfluid.2017.03.007>.

[34] P. Lallemand, L.-S. Luo, Theory of the lattice Boltzmann method: dispersion,
dissipation, isotropy, galilean invariance, and stability, Phys. Rev. E 61 (6)

[35] A. Bardow, I.V. Karlin, A.A. Gusev, General characteristic-based algorithm for

[36] X. Shan, Simulation of Rayleigh–Bénard convection using a lattice Boltzmann

[37] T. Zhang, B. Shi, Z. Guo, Z. Chai, J. Lu, General bounce-back scheme for
concentration boundary condition in the lattice-Boltzmann method, Phys. Rev.

[38] R. Krane, J. Jessee, Some detailed field measurements for a natural convection
flow in a vertical square enclosure, in: 1st ASME-JSME Thermal Engineering

convection velocity field in square and partitioned enclosures, in: Proceedings,

[40] S. Wakashima, T.S. Saitoh, Benchmark solutions for natural convection in a
cubic cavity using the high-order time–space method, Int. J. Heat Mass Transf.