Application of a coupled Lattice Boltzmann and Immersed Boundary Method for solving conjugated heat transfer problems

A Kasparek, P Łapka¹ and P Furmański
Institute of Heat Engineering, Warsaw University of Technology, 21/25 Nowowiejska St, 00-665 Warsaw, Poland
E-mail: plapka@itc.pw.edu.pl

Abstract. The paper presents development of a coupled Lattice Boltzmann and Immersed Boundary Method for solving conjugated heat and fluid flow problems in a domain with complex internal boundaries. The Lattice Boltzmann Method was applied to efficiently solve both Navier-Stokes and energy equations, while the Immersed Boundary Method accounted for presence of objects of complex shape inside the computational domain. The presented approach allowed to use fixed Cartesian grid on which complex internal boundaries were imposed. Several conjugated heat transfer and fluid flow problems in a domain with complex internal boundaries, i.e., laminar natural convection in a square cavity without and with internal cylinder and isothermal flow past stationary cylinder were analyzed. The results obtained were compared with the results presented in the literature successfully verifying the developed numerical simulator.

1. Introduction
In the recent years the Lattice Boltzmann Method (LBM) became very popular in fluid flow and heat transfer numerical simulations. Analyses with moving internal boundaries in both fluid-fluid and fluid-solid systems were also accounted for with this method [1]. The LBM was applied for simulations of moving vapour or gas bubbles in the liquid phase, moving droplets, fluid-particles interaction problems, fluid flow around very complex bodied, fluid flow in biological systems: blood flows in the cardiovascular system or insect flight, and solidification process in micro-, meso- and macro-scale [1-4]. The LBM is considered as the robust and computationally very efficient method with good solution accuracy and inherent features for parallelization. Additionally in this method solution of Poisson equation is avoided [1]. On the contrary in the classical continuum-based solutions of the Navier-Stokes equations (e.g.: SIMPLE [5] or fractional step [6] like methods) this equation is solved to find distribution of pressure or pressure correction in the computational domain. During such numerical simulations of conjugated fluid flow and heat transfer problems the convergence rate of the Poisson equation is usually very low leading to very time-consuming solutions.

The LBM utilise regular Cartesian grids similarly as the Immersed Boundary Method (IBM) [6, 7]. Therefore, both methods can be easily coupled and applied for simulations of fluid flow and heat transfer in domains containing many moving and evolving internal objects which may have very complex shapes. Simultaneously with calculation of velocity and temperature fields, tracking of the

¹ To whom any correspondence should be addressed.
exact location of interfaces between different phases or different media may be carried out. Evolution of bundle of equiaxed dendrites during micro/meso-scale modelling of solidification processes or large group of particles suspended and drifted by fluid are examples of such problems [2]. Applications of the Finite Volume or Finite Element Methods for calculation of distributions of velocity and temperature in such types of multi-object systems seems to be troublesome due to necessary frequent adaptation and regeneration of meshes and problems with solution convergence and stability as well as time consuming simulation [4].

Many coupled Lattice Boltzmann and Immersed Boundary approaches were developed and applied by several researchers to study fluid flow with heat transfer in complex domains. In the one of the first papers Jeong et al. [8] applied feedback forcing scheme to combine the IBM with the thermal LBM (TLBM) and simulated the natural convection around motionless bluff bodies in a square cavity. In turn Kang and Hassan [9] combined the IBM with two TLBM models (direct forcing approach) and solved convective heat transfer problems with not only stationary but also with moving boundaries – the natural convection in a square cavity with an eccentric cylinder and a cold particle sedimentation in an infinite channel. In the next paper Amiri Delouei et al. [10] studied non-Newtonian fluid flow over a heated cylinder applying the IBM-TLBM with direct forcing scheme. Successively, Huang and Wu [11] proposed the IBM-TLBM to simulate solid-liquid phase change problems without and with fluid flow. A novel IBM-TLBM to treat two kinds of temperature boundary conditions, i.e., the Dirichlet and Neumann types was developed by Hu et al. [12]. They investigated the 2D natural convection in a concentric horizontal annulus. Recently, Zhang et al. [13] developed a novel combined IBM-TLBM-Discrete Element Method scheme to simulate fluid-particle multiphase flows with heat transfer. All mentioned papers showed that the coupled IBM and LBM/TLBM is the promising approach to numerically study fluid flow and heat transfer in a domain with complex stationary or moving internal boundaries.

This paper presents development of the combined IBM and LBM/TLBM with direct-forcing scheme for simulation of fluid flow and heat transfer problems in domains with immersed internal bodies. The developed model was successfully verified by studying natural convection in a square cavity, isothermal flow around a cylinder and natural convection in a square enclosure with a heated cylinder.

2. Governing equations
The model was limited to 2D problems. A rectangular computational domain with or without stationary obstacle inside was considered. The incompressible Newtonian fluid with fixed thermophysical properties filled the analyzed cavity. Assuming laminar flow, the Navier-Stokes and energy equations were cast in the following form:

\[
\text{div} \mathbf{u} = 0 \quad (1)
\]

\[
\frac{\partial u_x}{\partial t} + \text{div}(u u_x) = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \text{div} (\text{grad} u_x) + S_x \quad (2)
\]

\[
\frac{\partial u_y}{\partial t} + \text{div}(u u_y) = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \text{div} (\text{grad} u_y) + S_y \quad (3)
\]

\[
\rho c_p \frac{\partial T}{\partial t} + \rho c_p \text{div} (u T) = \lambda \text{div} (\nabla T) \quad (4)
\]

where: \(c_p\) is specific heat, \(p\) – pressure, \(S\) – source term, \(t\) – time, \(T\) – temperature, \(u = [u_x, u_y]\) – velocity vector, \(x = [x, y]\) – Cartesian coordinates, \(\lambda\) – thermal conductivity, \(\nu\) – kinematic viscosity and \(\rho\) – density. The source terms were equal to: \(S_x = -\rho g \beta (T - T_{ref})\) and \(S_y = 0\) for the natural
convection and forced fluid flow, respectively. The symbol $g_i$ denotes $x$- or $y$- component of gravitational acceleration, $\beta$ – thermal expansion coefficient and $T_{\text{ref}}$ – reference temperature. The system of equations above was supplemented by the relevant boundary conditions. For the closed computational domain with and without obstacle (simulations of natural convection) all walls were assumed stationary with fixed temperature or zero heat flux (adiabatic). For the open cavity (simulations of flow past cylinder) the following conditions were assumed: fixed velocity at the inlet, outflow at the exit, symmetry at side walls and no-slip at the inner cylinder.

### 3. Lattice Boltzmann Method for fluid flow and heat transfer

#### 3.1. The LBM for fluid flow

The LBM fills the gap in CFD simulation between micro- and macro-scale by analyzing the behavior of the group of particles by using distribution functions $f_\alpha$ (meso-scale), where subscript $\alpha$ denotes discrete direction of the lattice. The core of the LBM is the collision and streaming (advection) of fluid particles. The lattice Boltzmann equation using the single relaxation time is following [14]:

$$f_\alpha (x + e_\alpha \Delta t, t + \Delta t) - f_\alpha (x, t) = \Omega_\alpha$$

(5)

Applying the Bhatnager, Gross and Krook (BGK) approach [15] to express the collision term $\Omega_\alpha$, the equation (5) reduces to:

$$f_\alpha (x + e_\alpha \Delta t, t + \Delta t) - f_\alpha (x, t) = -\frac{1}{\tau} [f_\alpha (x, t) - f_\alpha^{(eq)} (x, t)]$$

(6)

The D2Q9 lattice velocity model [9] with following discrete velocity vector was then applied:

$$e_\alpha = \begin{cases} 0 & \text{for } \alpha = 0 \\ c \left[ \cos \left( 0.5 (\alpha - 1) \pi \right), \sin \left( 0.5 (\alpha - 1) \pi \right) \right] & \text{for } \alpha = 1, 2, 3, 4 \\ \sqrt{2} c \left[ \cos \left( 0.5 (\alpha - 1) \pi \right), \sin \left( 0.5 (\alpha - 1) \pi \right) \right] & \text{for } \alpha = 5, 6, 7, 8 \end{cases}$$

(7)

where: $c = \Delta x / \Delta t$ is lattice speed, $\Delta x$ – lattice length, $\Delta t$ – time step and $\tau$ – relaxation time. The equilibrium distribution function for fluid flow, which appears in equation (6), was defined as follows [9]:

$$f_\alpha^{(eq)} = w_\alpha \rho (x, t) \left[ 1 + \frac{c_\alpha u}{c_s^2} + \frac{1}{2} \left( \frac{c_\alpha u}{c_s^2} \right)^2 - \frac{1}{2} \frac{u^2}{c_s^2} \right]$$

(8)

where: $c_s$ is lattice speed of sound defined as: $c_s = c/\sqrt{3}$ and $w_\alpha$ – the weighted coefficients equal to $w_0 = 4/9$ for $\alpha = 0, 1/9$ for $\alpha = 1-4$ and $1/36$ for $\alpha = 5-8$. The kinematic viscosity is calculated using Chaman-Enskog [16] formula: $\nu = c_s^2 \Delta t (\tau - 0.5)$. Density and velocity are defined respectively:

$$\rho = \sum_\alpha f_\alpha \quad \text{and} \quad \rho u = \sum_\alpha e_\alpha f_\alpha$$

(9)

The algorithm of the LBM is following:

- Calculation of collision step:

$$f_\alpha' (x, t) = f_\alpha (x, t) - \frac{1}{\tau} [f_\alpha (x, t) - f_\alpha^{(eq)} (x, t)]$$

(10)
Calculation of streaming step:

\[ f_a(x + e_\alpha \Delta t, t + \Delta t) = f'_a(x, t) \]  

Application of boundary conditions.

Calculation of densities and velocity vectors using equations (9).

3.2. The LBM with source term (force)

The source term (force) which is added to the equation (5) accounts for either buoyancy force or presence of internal boundary, i.e., in the present paper the IBM was combined with the LBM by applying the forcing approach (see subsection 3.4), in which the internal boundary is included by adding an external force to the equation (5). The split-forcing method proposed by Chen and Li [17] with some modifications was applied to deal with this source term. In this approach a half of discrete force was added to the distribution function before the collision step (the first-forcing step) and then after the collision step (the second-forcing step). The equation (5) with external force applying the split-forcing method takes the following form:

\[ f_a(x + e_\alpha \Delta t, t + \Delta t) - f'_a(x, t) = \]

\[ \frac{-1}{\tau} \left( f_a(x, t) - f^{(eq)}_a(x, t) \right) + \frac{\Delta t}{2} \left[ F_a(x, t) + F_a(x + e_\alpha \Delta t, t + \Delta t) \right] \]

and

\[ F_a = \frac{w_\alpha}{c_s^2} e_\alpha \mathbf{F}(x, t) \]

where \( \mathbf{F} \) is external force vector.

3.3. The LBM for thermal transport

The TLBM equation with the heat source term, used in the present paper, was following:

\[ g_a(x + e_\alpha \Delta t, t + \Delta t) - g_a(x, t) = \]

\[ \frac{-1}{\tau} \left( g_a(x, t) - g^{(eq)}_a(x, t) \right) + \frac{\Delta t}{2} \left[ Q_a(x, t) + Q_a(x + e_\alpha \Delta t, t + \Delta t) \right] \]

where: \( g_a \) is the distribution function for temperature in the \( \alpha \)-th discrete direction of the lattice and \( Q_a \) – heat source term defined as: \( Q_a = w_\alpha Q(x, t) \). The equilibrium function for temperature was similar to the equilibrium distribution function for the fluid flow:

\[ g^{(eq)}_a = w_\alpha T(x, t) \left[ 1 + \frac{e_\alpha \mathbf{u}}{c_s^2} + \frac{1}{2} \frac{(e_\alpha \mathbf{u})^2}{c_s^4} - \frac{1}{2} \frac{\mathbf{u}^2}{2 c_s^2} \right] \]

3.4. The combined IBM and LBM or TLBM

The IBM introduces additional internal boundary (interface) to existing regular grid at which velocity or thermal conditions should be satisfied. In this paper to combine the IBM and LBM or TLBM the direct-forcing scheme [3] was applied. The procedure for including internal boundaries starts after the streaming step and is following:

- The unforced velocities or temperatures (\( u^{nof} \) or \( T^{nof} \)) for next time step in all Cartesian grid nodes are calculated.
Using the 4-point discrete delta function velocities or temperatures on the internal boundary are approximated from unforced velocities or temperatures at nearby lattice points \([i, j]\) from following formulae [18]:

\[
\mathbf{u}^{\text{noF}}_b = \sum_{i,j} \mathbf{u}^{\text{noF}}_{i,j} D(x_{i,j} - \mathbf{x}_b)h^2 \quad \text{or} \quad T^{\text{noQ}}_b = \sum_{i,j} T^{\text{noF}}_{i,j} D(x_{i,j} - \mathbf{x}_b)h^2
\]

where: \(h\) is grid spacing (\(h = \Delta x\)), \(x_{i,j}\) – location of nearby lattice points and \(\mathbf{x}_b\) – location of the internal boundary. The function \(D\) is written as [18]:

\[
D(x_{i,j} - \mathbf{x}_b) = \frac{1}{h^2} d_h \left( \frac{x_{i,j} - \mathbf{x}_b}{h} \right) d_h \left( \frac{y_{i,j} - y_b}{h} \right)
\]

with

\[
d_h(r) = \begin{cases} 
\frac{1}{8} \left( 3 - |r| + \sqrt{1 + 4 |r| - 4r^2} \right) & 0 \leq |r| < 1 \\
\frac{1}{8} \left( 5 - 2 |r| - \sqrt{7 + 12 |r| - 4r^2} \right) & 1 \leq |r| < 2 \\
0 & |r| \geq 2
\end{cases}
\]

The external force or heat source term which account for presence of internal boundaries are obtained from following expressions [18]:

\[
F_b(x, t) = \rho(x, t) \frac{\mathbf{u}_b - \mathbf{u}^{\text{noF}}_b(x, t + \Delta t)}{\Delta t} \quad \text{or} \quad Q_b(x, t) = c_p \rho(x, t) \frac{T_b - T^{\text{noQ}}_b(x, t + \Delta t)}{\Delta t}
\]

where: \(\mathbf{u}^{\text{noF}}_b(x, t + \Delta t)\) or \(T^{\text{noQ}}_b(x, t + \Delta t)\) denote the tentative velocity or temperature, respectively at next time step and \(\mathbf{u}_b\) or \(T_b\) – velocity or temperature at the internal boundary, respectively.

Calculated external force or heat source term are distributed among neighboring nodes by using following expressions:

\[
F = \sum_b F_b(x - \mathbf{x}_b) \Delta s_b \quad \text{or} \quad Q = \sum_b Q_b(x - \mathbf{x}_b) \Delta s_b
\]

where: \(\Delta s_b\) is length of the segment between two neighboring points which define internal boundary. The calculated external forces or heat source terms in the Cartesian nodes are then included in the split-forcing approach – equation (12) or (14), respectively.

4. Results
Three cases: natural convection in a square cavity, isothermal flow past cylinder and natural convection in a square cavity with a cylinder were considered to verify proposed combined IBM and LBM or TLBM.

4.1. Case 1 – natural convection in a square cavity
The classical natural convective test in a domain without internal boundaries was used to check accuracy of LBM for coupled momentum and thermal transport. Computations were carried out until a steady-state solution was reached. The square computational domain of dimension \(L\) was divided into 200×200 uniform grid nodes with \(\Delta x = \Delta t = 1\). The Prandtl number was set to \(Pr = \nu/\alpha = 0.71\), where \(\alpha\) is the thermal diffusivity. The left and right walls were kept at constant dimensionless temperatures.
equal to $T_1 = 1$ and $T_2 = 0$, respectively. The top and bottom walls were adiabatic. For velocity no-slip boundary conditions at all walls were applied. The obtained results: dimensionless maximal $x$-velocity at $x/L = 0.5$ ($u^*_x$), dimensionless maximal $y$-velocity at $y/L = 0.5$ ($u^*_y$) and average Nusselt number at wall for different Rayleigh numbers ($Ra = \Delta TL^3 \rho \beta \Delta \alpha \nu$; where: $\beta$ is thermal expansion coefficient, $\Delta T$ – difference between maximum and minimum temperature in the domain) are presented in table 1 and additionally are compared with results obtained by other researchers [19, 20]. The averaged Nusselt number was defined as follows:

$$Nu_a = -\frac{1}{\Delta T} \int_0^L \frac{\partial T}{\partial x} \bigg|_{x=0} dy$$  \hspace{1cm} (21)

The obtained results match reference results very well – in most cases relative differences between the obtained and reference results are below 1%. Influence of the lattice length and time step on the obtained solution was also checked. The table 2 presents result for different lattice size and time step for Rayleigh number $Ra = 10^6$. Obtained results varies insignificantly with changes in lattice size and time step.

Table 1. Comparison of the dimensionless maximal $x$-velocity at the mid-width, dimensionless maximal $y$-velocity at the mid-height and average Nusselt number for different values of Rayleigh number (PM – present model).

| Ra    | Max $u^*_x$ at $x/L = 0.5$ | Max $u^*_y$ at $y/L = 0.5$ | $Nu_a$  |
|-------|-----------------------------|-----------------------------|---------|
| $10^3$| PM 3.63                     | 3.69                        | 1.13    |
|       | [19] 3.634                   | 3.679                       | 1.12    |
|       | [20] 3.6493                  | 3.6962                      | -       |
| $10^4$| PM 16.21                    | 19.62                       | 2.25    |
|       | [19] 16.2                    | 19.51                       | 2.243   |
|       | [20] 16.1798                 | 19.6177                     | -       |
| $10^5$| PM 35.06                    | 68.58                       | 4.53    |
|       | [19] 34.81                   | 68.22                       | 4.52    |
|       | [20] 37.7741                 | 68.6920                     | -       |
| $10^6$| PM 65.48                    | 220.0                       | 8.83    |
|       | [19] 65.33                   | 216.75                      | 8.8     |
|       | [20] 64.6912                 | 220.83                      | -       |

Table 2. Comparison of the dimensionless maximal $x$-velocity at the mid-width, dimensionless maximal $y$-velocity at the mid-height and average Nusselt number for $Ra = 10^6$ and for different values of lattice length and time step.

| $\Delta t$ | $\Delta x$ | Max $u^*_x$ at $x/L = 0.5$ | Max $u^*_y$ at $y/L = 0.5$ | $Nu_a$  |
|------------|------------|-----------------------------|-----------------------------|---------|
| 1.0        | 1.0        | 65.48                        | 220.00                      | 8.83    |
| 0.75       | 1.0        | 65.38                        | 219.91                      | 8.84    |
| 1.0        | 0.8        | 65.60                        | 220.54                      | 8.83    |
| 0.75       | 0.8        | 65.47                        | 220.43                      | 8.84    |
4.2. Case 2 – isothermal flow around a cylinder
The second analyzed problem was isothermal flow around stationary cylinder. It is a common benchmark for investigating the accuracy of the coupled IBM and LBM for different values of Reynolds number \( \text{Re} = \frac{2ru_\infty}{\nu} \), where: \( u_\infty \) is freestream velocity, \( r \) – radius of the cylinder. In this paper the Reynolds number was assumed \( \text{Re} = 40 \) which ensured steady state solution. The size of the computational domain was set to \( 40r \times 40r \). The cavity was discretized into uniform grid of following size: 800\( \times \)800 with \( \Delta x = \Delta t = 1 \). The center of the cylinder was located on the grid point of lattice coordinates equal to [200, 400]. Forcing points on the internal boundary were uniformly spaced \((\Delta s_b = h/1.5)\). At inlet the velocity was equal to \( u = [0.1, 0.0] \), while symmetry and outflow conditions were applied at side walls and exit from the domain, respectively. Figure 1 shows the streamlines near the internal cylinder for \( \text{Re} = 40 \) – the symmetric vortices are observed in the wake region just after the cylinder. Additionally the drag coefficient \( C_D \) and recirculation length (the size of flow recirculation region) \( L_w \) were estimated in the steady state and are compared with results presented in the literature – table 3. The drag coefficient was defined as follows:

\[
C_D = \frac{1}{u_\infty r} \sum_{b} F_b
\]  

The obtained results agree very well with reference results [18, 21, 22] – in most cases relative differences between obtained and reference results are below 1%. Additionally the developed method is able to capture the symmetric wake behind the cylinder in the steady state.

![Figure 1. Predicted streamlines around the cylinder for Re = 40.](image)

|       | \( C_D \) | \( L_w \) |
|-------|---------|---------|
| PM   | 1.602   | 2.502   |
| [18] | 1.597   | 2.525   |
| [21] | 1.580   | 2.490   |
| [22] | 1.565   | 2.310   |

Table 3. Comparison of the drag coefficient and recirculation length (the size of flow recirculation region) in the steady state for isothermal flow around cylinder with \( \text{Re} = 40 \) (PM – present model).
4.3. Case 3 – natural convection in a square enclosure with a cylinder

Natural convection in the square cavity enclosed by cold walls with hot inner cylinder located in a different position was the last analyzed case. The schematic view of the computational domain is presented in Figure 2. The diameter of inner cylinder was set to $D = 0.4L$, where: $L$ is the length of the square. Initially the cylinder was located in the center of the domain but its location varied along the vertical centerline in the range from $-0.25L$ to $0.25L$. The walls and cylinder were kept at dimensionless temperatures equal to 0 and 1, respectively. The fluid of Prandtl number equal to $Pr = \nu/\alpha = 0.71$ filled the domain and underwent natural convection with downward gravitational acceleration. The computational domain was divided into $400 \times 400$ grid nodes with $\Delta x = \Delta t = 1$. Forcing points on the cylinder were uniformly spaced with $\Delta s_b = h/1.5$.

The simulations were performed for three values of Rayleigh number: $Ra = 10^4$, $10^5$ and $10^6$ and for three positions of inner cylinder: the central location, $0.25L$ up from the center and $0.25L$ down from the center. Comparison of the average Nusselt numbers at cylinder for the steady state and for the central location of the cylinder are presented in table 4 and agree with results presented by Moukalled and Darwish [23] and Kim et al. [24] very well. The stream lines and isotherms for different Rayleigh numbers and positions of inner cylinder are presented in Figure 3. Increasing the Rayleigh number resulting in stronger wake region at the top of the square.

| $Ra$  | PM   | [23]  | [24] |
|-------|------|-------|------|
| $10^4$| 3.451| 3.331 | 3.414|
| $10^5$| 5.101| 5.080 | 5.139|
| $10^6$| 9.370| 9.374 | 9.390|

Figure 2. Square cavity with inner cylinder with dimensions and boundary conditions.
Figure 3. Predicted streamlines and isotherms for different Rayleigh numbers (Ra = 10^4, Ra = 10^5 and Ra = 10^6) and positions of inner cylinder: A) the central location, B) 0.25L up from the center and C) 0.25L down from the center.

5. Conclusions
The paper presents development of the combined IBM and LBM/TLBM with direct-forcing scheme for simulation of fluid flow and heat transfer. The proposed approach may deal with domains which contain stationary or moving internal objects of complex shapes using fixed Cartesian grid. The LBM
and TLBM were applied to efficiently solve Navier-Stokes and energy equations, respectively, while the IBM accounted for presence of objects of complex shape inside the computational domain.

At first the formulation of the proposed numerical model was given. Then three cases were analysed i.e.: natural convection in a square cavity, isothermal flow past cylinder and natural convection in a square cavity with inner hot cylinder. The results obtained were compared with the reference values, successfully verifying the developed approach. For majority of the analyzed cases relative difference between obtained results and results presented in the literature were below 1%. The future work will focus on coupling of the presented numerical model with the IBM for radiative heat transfer recently developed by Łapka and Furmański [6, 25-28] to effectively solve conjugated conductive/convective-radiative problems.

References
[1] Inamuro T 2012 Fluid Dyn. Res. 44 024001
[2] Feng Z-G and Michaelides E E 2004 J. Comput. Phys. 195 602
[3] Feng Z-G and Michaelides E E 2005 J. Comput. Phys. 202 20
[4] Sun D, Zhu M, Pan S, Yang C R and Raabe D 2011 Comput. Math. Appl. 61 3585
[5] Versteeg H K and Malalasekera W 2007 An Introduction to Computational Fluid Dynamics (Harlow: Pearson Education Ltd)
[6] Łapka P and Furmański P 2012 Int. J. Heat Mass Tran. 55 4941
[7] Mittal R and Iaccarino G 2005 Annu. Rev. Fluid Mech. 387 239
[8] Jeong H K, Yoon H S, Ha M Y and Tsutahara M 2010 J. Comput. Phys. 229 2526
[10] Amiri Delouei A, Nazari M, Kayhani M H and Succi S 2014 Phys. Rev. E Stat. Nonlin. Soft Matter Phys. 89 053312
[11] Huang R and Wu H 2014 J. Comput. Phys. 277 305
[12] Hu Y, Li D, Shu S and Niu X 2015 Int. J. Heat Mass Tran. 81 591
[13] Zhang H, Yu A, Zhong W and Tan Y Int. J. Heat Mass Tran. 91 12334
[14] Chen S Y, Wang Z, Shan X W and Doolen G D 1992 J Stat. Phys. 68 379
[15] Bhatnagar P L, Gross E P and Krook M A 1954 Phys. Rev. 94 511
[16] Guo Z L, Zheng C G and Shi B C 2002 Phys. Rev. E 65 046308
[17] Cheng Y G and Li J P 2008 Int. J. Numer. Meth. Fl. 56 629
[18] Kang S K and Hassan Y A 2011 Int. J. Numer. Meth. Fl. 66 1132
[19] de Vahl Davis G 1983 Int. J. Numer. Meth. Fl. 3 249
[20] Mayne D A, Usmani A S and Crapper M 2000 Int. J. Numer. Method H. 10 598
[21] Le D V, Khoo B C and Lim K M 2008 Comput. Method. Appl. M. 197 2119
[22] Wu J and Shu C 2009 J. Comput. Phys. 228 1963
[23] Moukalled F and Acharya S 1996 J. Thermophys Heat Transfer 10 524
[24] Kim B, Lee D, Ha M and Yoon H 2008 Int. J. Heat Mass Tran. 51 1888
[25] Łapka P and Furmański P 2010 J. Heat Tran. 132 023504
[26] Łapka P and Furmański P 2011 AIP Conf. Proc. 1389 159
[27] Łapka P and Furmański P 2012 Int. J. Heat Mass Tran. 55 4953
[28] Seredyński M, Łapka P, Banaszek J and Furmański P 2015 Int. J. Heat Mass Transf; 90 790

Acknowledgments
The authors are grateful to the Polish Ministry of Science and Higher Education for funding from its Grant No. IP 2012 021172.