

Entropic multirelaxation-time lattice Boltzmann method for moving and deforming geometries in three dimensions

B. Dorschner,^{*} S. S. Chikatamarla,[†] and I. V. Karlin[‡]*Aerothermochemistry and Combustion Systems Lab, Department of Mechanical and Process Engineering, ETH Zurich, CH-8092 Zurich, Switzerland*

(Received 27 March 2017; published 13 June 2017)

Entropic lattice Boltzmann methods have been developed to alleviate intrinsic stability issues of lattice Boltzmann models for under-resolved simulations. Its reliability in combination with moving objects was established for various laminar benchmark flows in two dimensions in our previous work [B. Dorschner, S. Chikatamarla, F. Bösch, and I. Karlin, *J. Comput. Phys.* **295**, 340 (2015)] as well as for three-dimensional one-way coupled simulations of engine-type geometries in B. Dorschner, F. Bösch, S. Chikatamarla, K. Boulouchos, and I. Karlin [*J. Fluid Mech.* **801**, 623 (2016)] for flat moving walls. The present contribution aims to fully exploit the advantages of entropic lattice Boltzmann models in terms of stability and accuracy and extends the methodology to three-dimensional cases, including two-way coupling between fluid and structure and then turbulence and deforming geometries. To cover this wide range of applications, the classical benchmark of a sedimenting sphere is chosen first to validate the general two-way coupling algorithm. Increasing the complexity, we subsequently consider the simulation of a plunging SD7003 airfoil in the transitional regime at a Reynolds number of $Re = 40\,000$ and, finally, to access the model's performance for deforming geometries, we conduct a two-way coupled simulation of a self-propelled anguilliform swimmer. These simulations confirm the viability of the new fluid-structure interaction lattice Boltzmann algorithm to simulate flows of engineering relevance.

DOI: [10.1103/PhysRevE.95.063306](https://doi.org/10.1103/PhysRevE.95.063306)

I. INTRODUCTION

Flows induced by complex and moving geometries are of paramount interest in many fields ranging from industrial applications for the optimization of internal combustion engines, turbines, etc., to flows in the field of biofluid mechanics in order to enhance our basic understanding of the propulsion mechanisms of animals, the flow through the cardiovascular system, and many more (see, e.g., Howell *et al.* [1], Schmitt *et al.* [2], Kern and Koumoutsakos [3], De Tullio *et al.* [4], Randles [5]). Besides the valuable insight from experimental studies, light may be shed on these complex mechanisms through numerical simulations. Thus, accuracy, robustness, and efficiency of the numerical treatment is crucial.

In this context, the lattice Boltzmann method (LBM) has gained increasing attention due to its accuracy and reliability in various regimes, including turbulence, thermal flows, compressible flows, micro flows, porous media and multiphase flows among many others (see Ansumali *et al.* [6], Bösch *et al.* [7], Frapolli *et al.* [8,9], Mazloomi *et al.* [10], and Mendoza *et al.* [11]). The LBM originates from kinetic theory and describes the flow in terms of discretized particle distribution functions (populations) $f_i(\mathbf{x}, t)$ associated with a set of discrete velocities $\mathbf{c}_i, i = 1, \dots, Q$, where the dynamics of the populations are constructed to recover the desired macroscopic equations in the hydrodynamic limit. Efficiency is achieved by organizing the discrete velocities into a regular lattice, resulting in a stream-and-collide algorithm with exact propagation and spatially local nonlinearity implemented through the collision operator. The first realization, which

made a practical connection of LBM to fluid dynamics, was the well-known lattice Bhatnagar-Gross-Krook (LBGK) model. The LBGK model remains popular due to its simplicity and numerical efficiency. Despite these attractive properties, the LBM was not competitive for long due to the lack of numerical stability and accurate boundary conditions in the simulation of high Reynolds number flows. This promoted the development of various models intended to resolve this issue, including explicit turbulence models in LBM formulation, crystallographic, regularized as well as multiple-relaxation time (MRT) lattice Boltzmann models among others (see, e.g., D'Humières [12], Karlin *et al.* [13], Chen *et al.* [14], Latt and Chopard [15], and Namburi *et al.* [16]). Notable is the entropic lattice Boltzmann method (Karlin *et al.* [13]), which, by introduction of a discrete entropy function H as the determining factor of the relaxation, features nonlinear stability. In contrast to LBGK, the relaxation parameter is chosen adaptively at each point in space and time to locally ensure the discrete-time H-theorem. Consequently, an effective viscosity is prescribed by the second law of thermodynamics, which may both smoothen or enhance the flow field. Recently, the entropic concept was extended to multirelaxation time models (KBC models), where the additional freedom of multiple relaxation parameters is used to keep the effective viscosity at its nominal value while retaining stability. Excellent performance of KBC models in terms of stability and accuracy for under-resolved simulations have been reported in the works of Karlin *et al.* [17], Bösch *et al.* [7], and Dorschner *et al.* [18]. Due to its advantages, all simulations in the remainder of the paper are conducted using the KBC model with the focus on low Reynolds number and transitional flow regime.

In order to establish these entropy-based lattice Boltzmann models as a robust and predictive tool for complex engineering problems, the topic of boundary conditions needs to be addressed, as its implementation is crucial for accuracy and

^{*}benedikt.dorschner@lav.mavt.ethz.ch[†]chikatamarla@lav.mavt.ethz.ch[‡]Corresponding author: karlin@lav.mavt.ethz.ch

stability of the entire simulation. Various proposals for flat, curved, and moving boundaries can be found in the literature. Most popular for its simplicity is the bounce-back boundary condition, where the geometry is represented using a staircase approximation. For a more accurate representation of the geometry, common approaches use interpolation or extrapolation of the populations to account for the curvature of the object. However, spurious shocks for turbulence simulation at the boundary limit their usage to resolved and moderate Reynolds number flows. An interesting realization of the bounce-back can be found for a recent, so-called crystallographic LBM, in Ref. [16]. Another common approach for complex moving objects is the immersed boundary method (IBM), which was originally proposed by Peskin [19] for the simulation of blood flow. Notable in the LBM realm is the IBM procedure as introduced by Suzuki and Inamuro [20], which resolved the common issue of streamlines penetrating the object and more accurately satisfies the no-slip boundary condition.

In our recent contribution [21], we aimed to resolve these issues by employing Grad's approximation to impose the desired boundary conditions on the slow varying moments rather than the fluctuating population directly. In Dorschner *et al.* [21] excellent stability properties and second-order accuracy were reported. Further, this boundary condition was validated for various laminar benchmark simulations with stationary and moving walls in two dimensions. In this paper we extend these models for moving objects in three dimensions with a successively increasing level of complexity. First, we consider the classical two-way coupled benchmark simulation of a falling sphere under gravity to validate the boundary condition and the two-way coupling algorithm. The next level of complexity is a simulation of a plunging airfoil at $Re = 40\,000$ in the transitional regime, where we report a detailed comparison with the experimental and numerical studies of McGowan *et al.* [22] and Visbal [23], respectively. Finally, the performance for deforming geometries is accessed on the example of a self-propelled anguilliform swimmer.

The paper is organized as follows: In Sec. II, the entropic multirelaxation time model is reviewed followed by a brief exposure of the boundary conditions in Sec. III. Subsequently, in Sec. IV, we present our results for the simulation of the sedimenting sphere, the plunging airfoil, and the anguilliform swimmer. Finally, concluding remarks are given in Sec. V.

II. ENTROPIC MULTIRELAXATION TIME LATTICE BOLTZMANN MODEL—KBC MODEL

The population $f_i(\mathbf{x}, t)$ is evolved in accordance with the discrete kinetic equation

$$f_i(\mathbf{x} + \mathbf{c}_i, t + 1) = f'_i = (1 - \beta)f_i(\mathbf{x}, t) + \beta f_i^{\text{mirr}}(\mathbf{x}, t), \quad (1)$$

where the streaming step is prescribed by the left-hand side and the postcollision state f'_i is given on the right-hand side by a convex-linear combination of $f_i(\mathbf{x}, t)$ and the mirror state $f_i^{\text{mirr}}(\mathbf{x}, t)$. Various lattice Boltzmann schemes differ in the way the mirror state is constructed. The KBC model as employed in this paper represents the population f_i in its natural moment basis and partitions it into three parts as

$$f_i = k_i + s_i + h_i, \quad (2)$$

where the kinematic part k_i depends only on the locally conserved moments. The shear part is denoted by s_i and includes the deviatoric stress tensor. All remaining higher-order moments not included in k_i or s_i are lumped into h_i . The mirror state, in Eq. (1), can now be expressed as

$$f_i^{\text{mirr}} = k_i + (2s_i^{\text{eq}} - s_i) + [(1 - \gamma)h_i + \gamma h_i^{\text{eq}}], \quad (3)$$

where s_i^{eq} and h_i^{eq} denote s_i and h_i evaluated at equilibrium and the parameter γ quantifies the relaxation rate of the higher-order moments.

In the athermal case, the equilibrium populations f_i^{eq} are obtained by minimization of the discrete entropy function H subject to local mass and momentum conservation,

$$\min \left\{ H(f) = \sum_{i=1}^Q f_i \ln \left(\frac{f_i}{W_i} \right) \right\}, \quad \text{s.t.} \sum_{i=1}^Q \{1, \mathbf{c}_i\} f_i = \{\rho, \mathbf{j}\}, \quad (4)$$

where the lattice-specific weights are denoted by W_i . In this paper the standard three-dimensional $D3Q27$ lattice with a speed of sound of $c_s = 1/\sqrt{3}$ is used for all computations, where D and Q are the dimensionality and the number of discrete velocities, respectively. The exact solution to the minimization problem can be found in Ansumali *et al.* [24] (see also Bösch *et al.* [7] for a detailed discussion).

The Chapman-Enskog analysis applied to this system yields the Navier-Stokes equations with the shear viscosity ν and the bulk viscosity ξ as

$$\nu = c_s^2 \left(\frac{1}{2\beta} - \frac{1}{2} \right), \quad \xi = c_s^2 \left(\frac{1}{\gamma\beta} - \frac{1}{2} \right). \quad (5)$$

As for any MRT model trying to increase stability, the KBC model exploits the fact that the higher-order moments may be relaxed independently without affecting the hydrodynamic limit. However, unlike in the conventional MRT approach, the relaxation parameter γ in the KBC model is not a constant tuning parameter but is rather computed locally in every time step and at every grid point by minimizing the discrete entropy function in the postcollision state f'_i . The minimization results in the following equation for the stabilizer γ :

$$\sum_{i=1}^Q \Delta h_i \ln \left[1 + \frac{(1 - \beta\gamma)\Delta h_i - (2\beta - 1)\Delta s_i}{f_i^{\text{eq}}} \right] = 0, \quad (6)$$

where $\Delta s_i = s_i - s_i^{\text{eq}}$ and $\Delta h_i = h_i - h_i^{\text{eq}}$. Moreover, an analytic approximation for the stabilizer γ may be obtained by expansion of Eq. (6) to the first nonvanishing order of $\Delta s_i/f_i^{\text{eq}}$ and $\Delta h_i/f_i^{\text{eq}}$ as

$$\gamma = \frac{1}{\beta} - \left(2 - \frac{1}{\beta} \right) \frac{\langle \Delta s | \Delta h \rangle}{\langle \Delta h | \Delta h \rangle}, \quad (7)$$

where the entropic scalar product $\langle X | Y \rangle = \sum_{i=1}^Q (X_i Y_i / f_i^{\text{eq}})$ is introduced to ease notation. The accuracy of this approximation has proven to be sufficient for all simulations reported in this paper. For an in-depth discussion on the entropic multirelaxation time models, the reader is referred to Bösch *et al.* [7].

III. WALL-BOUNDARY CONDITIONS

We here aim at extending the capabilities of KBC models to an accurate implementation of moving and deforming objects. Thus, the boundary conditions as detailed in Dorschner *et al.* [21] are briefly summarized.

In order to complete the streaming step in the LB algorithm, the set of populations $\bar{D}$ advected from a solid node $\mathbf{x}_s$ to a fluid boundary node $\mathbf{x}_b$ are unknown and need to be specified in accordance with the desired boundary conditions. In Dorschner *et al.* [21], we proposed to approximate these “missing” populations by an analog of Grad’s distribution function. This results in a parametrization of the distribution in terms of relevant moments, where not only locally conserved but also other pertinent moments may be taken into account. It is worth noting that this is in line with the notion of minimal entropy as used in bulk for the KBC model as the Grad distribution may analogously be derived using minimum entropy or quasiequilibrium considerations as discussed in Gorban and Karlin [25]. In the athermal case as considered in this paper, it suffices to include the locally conserved quantities along with the pressure tensor $\mathbf{\Pi}$. Explicitly, the Grad distribution with those contributions reads

$$f_i^*(\rho, \mathbf{u}, \mathbf{\Pi}) = W_i \left[\rho + \frac{\rho}{c_s^2} \mathbf{c}_i \cdot \mathbf{u} + \frac{1}{2c_s^4} (\mathbf{\Pi} - \rho c_s^2 \mathbf{I}) : (\mathbf{c}_i \otimes \mathbf{c}_i - c_s^2 \mathbf{I}) \right], \quad (8)$$

where the pressure tensor is approximated as

$$\mathbf{\Pi} = \mathbf{\Pi}^{\text{eq}} + \mathbf{\Pi}^{\text{neq}}, \quad (9)$$

with

$$\mathbf{\Pi}^{\text{eq}} = \rho c_s^2 \mathbf{I} + \rho \mathbf{u} \otimes \mathbf{u}, \quad (10)$$

$$\mathbf{\Pi}^{\text{neq}} = -\frac{\rho c_s^2}{2\beta} (\nabla \mathbf{u} + \nabla \mathbf{u}^\dagger). \quad (11)$$

The full specification of Grad’s distribution requires the density ρ , the velocity $\mathbf{u}$, and the pressure tensor to be prescribed. For this purpose, the concept of target values is introduced. The momentum exerted by the object is accounted for by specifying an appropriate target velocity $\mathbf{u}_{\text{tgt}}$ at $\mathbf{x}_b$, which may be obtained by an interpolation involving the wall velocity $\mathbf{u}_{w,i} = \mathbf{u}(\mathbf{x}_{w,i}, t)$ at the intersection point $\mathbf{x}_{w,i}$ with the object along the velocity vector $\mathbf{c}_i$ and the velocities $\mathbf{u}_{f,i} = \mathbf{u}(\mathbf{x}_{f,i}, t)$ at the adjacent fluid nodes $\mathbf{x}_{f,i} = \mathbf{x}_b + \mathbf{c}_i \delta_t$ for $i \in \bar{D}$. Using an averaged linear interpolation for the target velocity yields

$$\mathbf{u}_{\text{tgt}} = \frac{1}{n_{\bar{D}}} \sum_{i \in \bar{D}} \frac{q_i \mathbf{u}_{f,i} + \mathbf{u}_{w,i}}{1 + q_i}, \quad (12)$$

where $n_{\bar{D}}$ is the number of unknown populations and $q_i = \|\mathbf{x}_b - \mathbf{x}_{w,i}\| / \|\mathbf{c}_i\|$. The target density, on the other hand, has two contributions corresponding to the static and the dynamic part as

$$\rho_{\text{tgt}} = \rho_{\text{stat}} + \rho_{\text{dyn}} \quad (13)$$

with

$$\rho_{\text{stat}} = \sum_{i \in \bar{D}} f_i^{\text{bb}} + \sum_{i \notin \bar{D}} f_i, \quad (14)$$

$$\rho_{\text{dyn}} = \sum_{i \in \bar{D}} 2W_i \rho_b \frac{\mathbf{c}_i \cdot \mathbf{u}_{w,i}}{c_s^2}, \quad (15)$$

where the static part ρ_{stat} is the implied density if one were to use the bounce-back boundary condition to ensure no mass flux through the boundary. The reflected population f_i^{bb} is defined as $f_i^{\text{bb}} = \tilde{f}_i$, where $\tilde{f}_i$ is associated with the velocity vector $\tilde{\mathbf{c}}_i = -\mathbf{c}_i$. The dynamic part ρ_{dyn} accounts for the density alteration caused by the mass displacement by the moving body and $\rho_b = \rho(\mathbf{x}_b, t)$ denotes the postcollision density at the boundary node. The dynamic density may be derived by introducing a forcing term F_i , which is necessary for the displacement. The mass and momentum conservation $\sum_{i=1}^Q F_i = 0$ and $\sum_{i=1}^Q \mathbf{c}_i F_i = \rho_b \mathbf{u}_w$ directly lead to

$$F_i = 2W_i \rho_b \frac{\mathbf{c}_i \cdot \mathbf{u}_w}{c_s^2} \quad (16)$$

for the $D3Q27$ lattice, where the summation over all unknown populations in $\bar{D}$ yields the implied change in density for moving objects. Finally, the pressure tensor is prescribed by computing $\nabla \mathbf{u}$ using a finite difference scheme and the velocity values from the previous time step and evaluating Eqs. (10) and (11).

In particular, the velocity gradient along the x axis and similarly for the y and z coordinates may be approximated as

$$\left. \frac{\partial \mathbf{u}}{\partial x} \right|_{\mathbf{x}=\mathbf{x}_b} = \frac{\mathbf{u}(\mathbf{x}_f) - \mathbf{u}(\mathbf{x}_w)}{x_f - x_w}, \quad (17)$$

where x_w, x_f denote the location of the wall intersection and the next fluid node (with respect to the boundary node $\mathbf{x}_b$) along the x axis.

Another aspect to be considered for moving objects is the reinitialization or refill of the lattice sites, which are uncovered as the objects passes by. For such nodes, we again employ the Grad distribution as given in Eq. (8) with the wall velocity and a local density average, which has shown to exhibit reasonably small pressure oscillations (see also Ref. [26]). The pressure tensor is evaluated in the same manner as for the boundary conditions.

These boundary conditions constitute the action of the object onto the fluid. On the other hand, for two-way coupled simulations, the feedback exerted from the fluid onto the object is accounted for by solving Newton’s equations of motion and will be detailed in the corresponding section below.

IV. RESULTS AND DISCUSSION

A. Sedimenting sphere

As a first step to validate the proposed two-way coupled KBC algorithm, we consider the classical benchmark of a settling particle under gravity. Note that this case has previously been studied in Ref. [18] but is again presented to provide a self-contained validation. For this purpose, we conduct two simulations. In the first simulation, we keep the sphere stationary and impose a mean flow in order to validate

TABLE I. Flow past a sphere at $Re = 100$.

Contribution	C_d	L/D_s
Johnson and Patel [27]	1.1	0.88
Eitel-Amor <i>et al.</i> [28]	1.098	0.87
Present	1.1	0.86

and compute the drag coefficient and the recirculation length for a Reynolds number of $Re = u_\infty D_s / \nu = 100$. The sphere is resolved with $D_s = 30$ grid points and the characteristic velocity $u_\infty = 0.01$ is given by the mean flow velocity. After the initial transient, the drag coefficient C_d and the recirculation length L are measured and the results are listed in Table I along with literature values. With a good agreement to all reference data, we perform a second simulation for which there is no mean flow velocity but instead the sphere is settling under gravity. When released in a quiescent fluid with density ρ_f , the sphere with density ρ_s accelerates towards its terminal settling velocity u_t for which the gravitational force $F_g = \pi D_s^3 \rho_s g / 6$ is balanced by the buoyancy force $F_b = \pi D_s^3 \rho_f g / 6$ and the drag force $F_d = \rho_f u^2 \pi D_s^2 C_d / 8$. Establishing the force balance relates the density ratio to the terminal velocity as

$$\frac{\rho_s}{\rho_f} = 1 + \frac{3u_t^2 C_d}{4D_s g}. \quad (18)$$

In this two-way coupled simulation, the feedback from the fluid onto the sphere is prescribed by Newton's equations for the particle motion as

$$\dot{\mathbf{x}}_s = \mathbf{u}_s, \quad (19)$$

$$\ddot{\mathbf{x}}_s = \frac{\mathbf{F}}{m_s} + \left(1 - \frac{\rho_f}{\rho_s}\right) \mathbf{g}, \quad (20)$$

which is solved by a Euler integration and the force $\mathbf{F}$ is computed via the Galilean invariant momentum exchange method (see Wen *et al.* [29]). The evolution of the settling velocity is shown in Fig. 1 for an imposed drag coefficient from the stationary simulation and a nominal settling velocity of $u_{t,n} = 0.01$. After the initial acceleration, a terminal

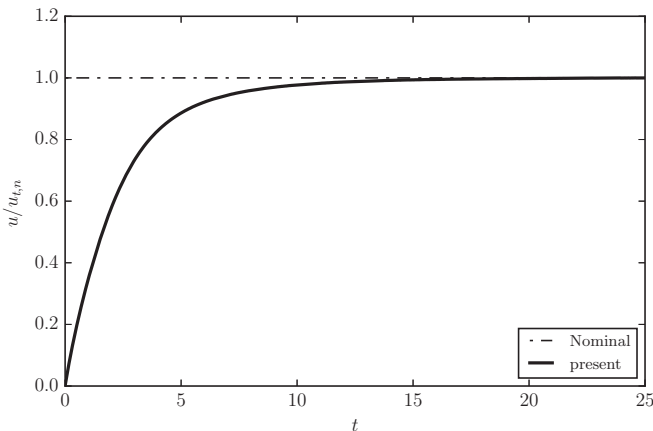


FIG. 1. Temporal velocity evolution of a sedimenting sphere.

TABLE II. Terminal settling velocity for a sedimenting sphere at $Re = 100$.

Contribution	$u_t/u_{t,n}$
Kern and Koumoutsakos [3]	1.006
Li <i>et al.</i> [30]	1.005
Present	1.0003

velocity of $u_t = 0.0100033$ is reached at $t = t_{lb} u_{t,n} / D = 25$ nondimensional time units, which corresponds to less than 0.033% error (see also Table II for comparison with literature values). Thus, Galilean invariance is established between the stationary and the moving case and therefore validates the basic two-way coupling algorithm.

B. Plunging airfoil

In this section, we leave the laminar flow regime and focus on the transitional flow past a plunging airfoil at a Reynolds number $Re = 40\,000$. Motivated to deepen our understanding of the complex physics relevant to small fliers, small unmanned air vehicles, micro air vehicles and so on, this setup was recently investigated experimentally and numerically in the works of McGowan *et al.* [22], Visbal [23], and Ou *et al.* [31]. Analogous to small fliers, the flow is mainly characterized by the formation of dynamic-stall vortices on the leading edge due to the large induced angle of attack. The transitional flow regime is particularly demanding for turbulence models as a high-Reynolds number analysis may not be valid in the presence of both laminar and turbulent flow. This gives us the opportunity to test both the KBC model and the implementation of moving boundaries to its full extent. Notable is the recent study from Visbal [23] using an implicit large-eddy simulation (ILES, see, e.g., Margolin and Rider [32], Margolin *et al.* [33] for details and the rationale of ILES) with high-order compact schemes for the spatial derivatives needed to capture the transitional processes accurately and a Pade-type low-pass filter to gain stability.

In particular and as shown in Fig. 2, the flow past a plunging SD7003 airfoil with a static angle of attack $\alpha_0 = 4^\circ$ is considered in this contribution. The airfoil is resolved by $L = 400$ lattice points using two levels of block refinement with a refinement ratio of two near the airfoil and a domain of $[10L \times 5L \times 0.2c]$ in the streamwise, lateral, and spanwise directions, respectively. The grid refinement technique used within this context is detailed and validated in Dorschner *et al.* [34]. As in Visbal [23], a sinusoidal plunging motion

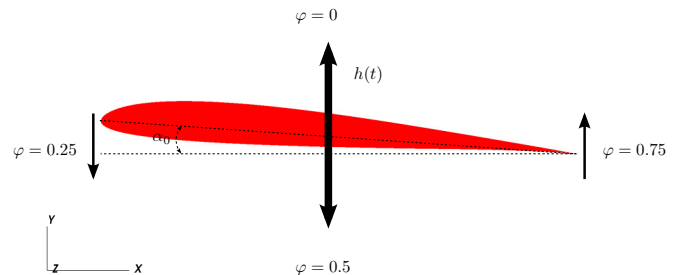


FIG. 2. Schematic of a plunging airfoil.

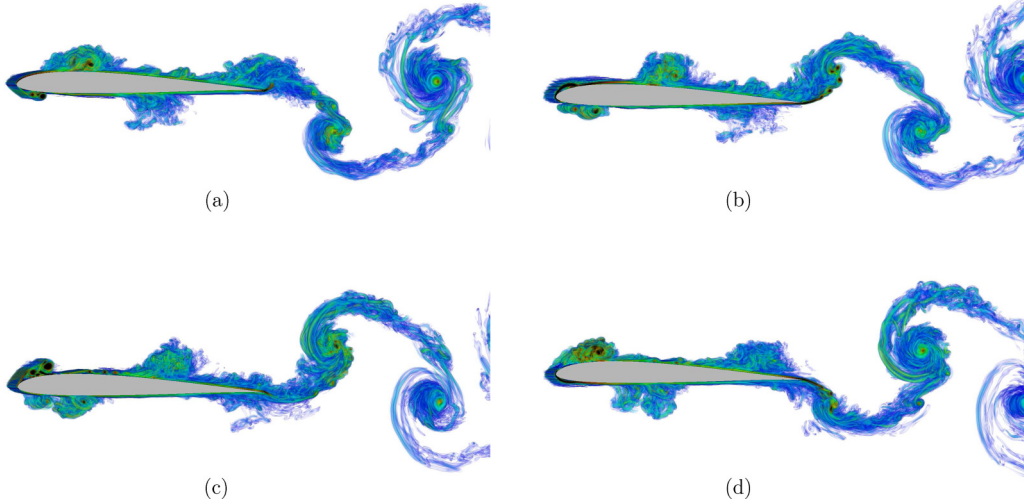


FIG. 3. Volume rendering of vorticity for various phases of the plunging airfoil. (a) $\varphi = 0$. (b) $\varphi = 0.25$. (c) $\varphi = 0.5$. (d) $\varphi = 0.75$.

of the airfoil is prescribed with a plunging amplitude of $h_D = 0.05c$ and a reduced frequency of $k = \pi f c / u_\infty = 3.93$ as

$$h(t) = \frac{h_D}{c} \sin[2kF(t)], \quad (21)$$

where

$$F(t) = 1 - e^{at}, \quad a = 4.6/2 \quad (22)$$

is an initial delay function to achieve a smooth transition from the resting airfoil to the plunging motion and the time t_{lb} in lattice units is nondimensionalized as $t = t_{lb} u_\infty / c$. This corresponds to an induced angle of attack $\alpha = 21.5^\circ$, which is sufficient for the formation of unsteady leading-edge separation and dynamic-stall-like vortices.

Capturing the main flow features occurring during the periodical motion of the airfoil, Fig. 3 shows four volume renderings of the instantaneous vorticity for phase angles of $\varphi = \{0, \frac{1}{4}, \frac{1}{2}, \frac{3}{4}\}$, corresponding to maximum upward displacement, maximum downward velocity, maximum downward displacement, and maximum upward velocity, respectively (see also Fig. 2). At the beginning of a new cycle, at a position of maximum upward displacement, the boundary layer near the leading-edge appears laminar and attached to the surface [see Fig. 3(a)]. At later times, caused by the downward acceleration, an emerging flow separation can be observed at the leading edge [see Fig. 3(b)]. At bottom dead center (BDC), the flow is fully separated at the leading-edge on the upper surface of the airfoil, causing the formation of two coherent vortices [see Fig. 3(c)]. However, due to spanwise instabilities, these vortices break down into three-dimensional, fine-scale turbulence during the subsequent upward acceleration [see Fig. 3(d)]. While diffusing and annihilating, these vortices propagate close to the airfoil surface during the following cycle. Note that due to the high frequency, a new pair of vortices is formed before the vortex pair of the previous cycle is shed from the trailing edge. Identical but less pronounced is the flow structure on the lower surface of the airfoil, where the large negative motion-induced angle of attack causes the formation of two coherent vortex structures, which subsequently break

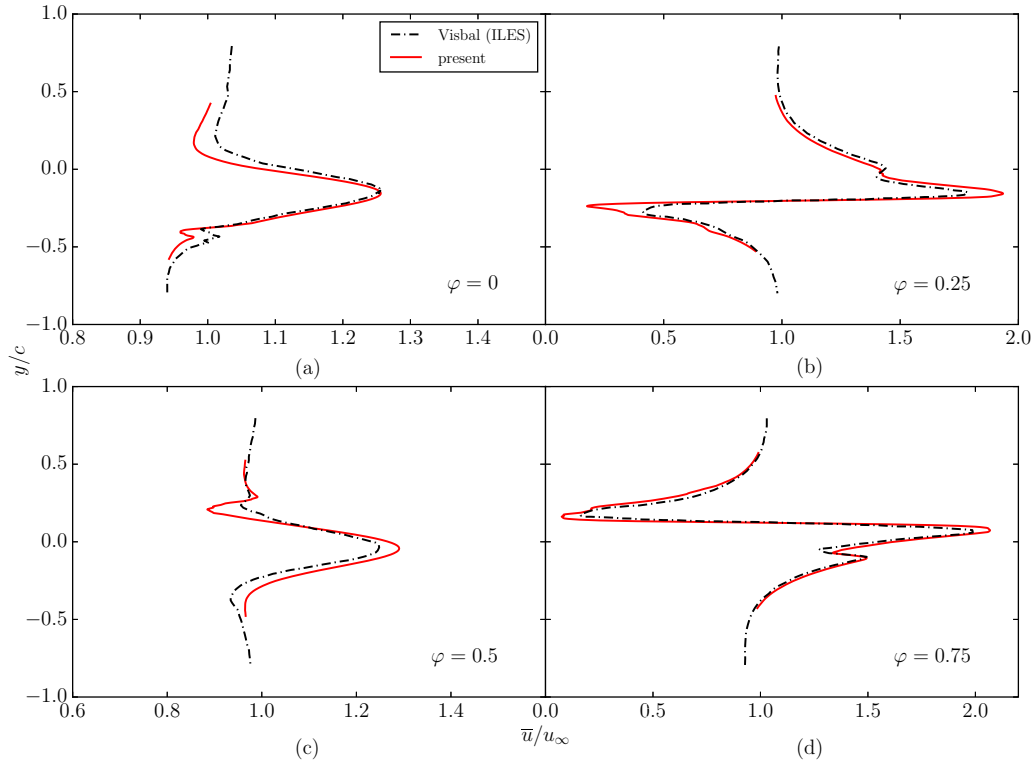
down into turbulence. Similar observations were reported by Visbal [23].

More quantitatively, in Fig. 4, we compare the phase-averaged velocity profiles in the near wake of the airfoil at $x/c = 1.5$ with the study of Visbal [23]. In total 25 cycles were computed and the first 10 were neglected in the accumulation of statistics to avoid accounting for the initial transient. It is apparent that the main flow features are captured for both simulations and the agreement is good. The location of the minimum and maximum of the flow velocity agree well but differ slightly in terms of magnitude. The present simulation exhibits larger velocity magnitudes in comparison with the more smoothed ILES simulation. In addition, a comparison at a location further downstream at $x/c = 2$ with an experimental investigation using Particle Image Velocimetry (PIV), an immersed boundary method, and the NASA-CFL3D code from the study of McGowan *et al.* [22] is shown in Fig. 5. It is apparent that due to the complexity of the problem all data have notable discrepancies but the agreement of the present study with the experiment is reasonable.

Finally, we compare the evolution of the aerodynamic forces for three selected cycles with Visbal [23] in Fig. 6. The lift coefficient is defined as $C_L = F_L / (1/2 \rho u_\infty^2 c L_s)$, where F_L is the total lift force acting on the airfoil and L_s is the extent in spanwise direction. Analogously, the drag coefficient is given by $C_D = F_D / (1/2 \rho u_\infty^2 c L_s)$, where the drag force is denoted by F_D . As shown in Fig. 6, the lift coefficient C_L is dominated by the prescribed motion of the airfoil and has a large amplitude. On the other hand, the amplitude of the drag coefficient is much smaller but notably there is a net mean thrust corresponding to a mainly negative drag coefficient. The comparison with the ILES is excellent.

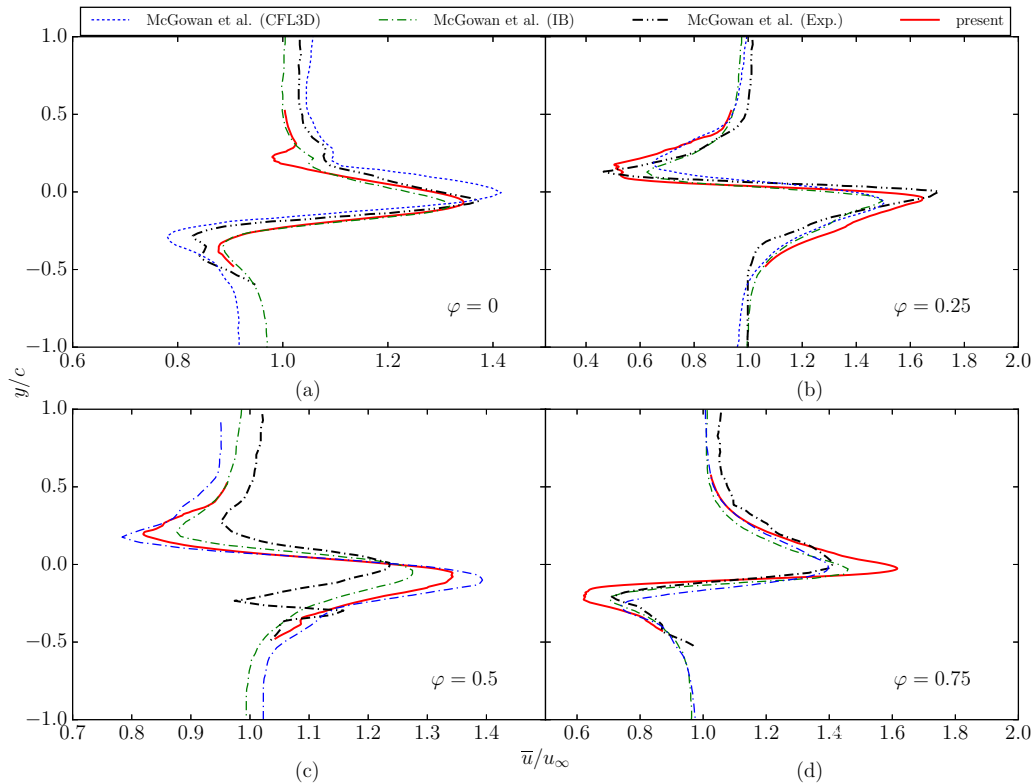
C. Anguilliform swimmer

In the field of biofluid mechanics, the topic of aquatic animal propulsion mechanisms is much discussed among biologists, neuroscientist as well as engineers trying to mimic these mechanisms to increase efficiency in technical applications (see, e.g., Ekeberg [35], Ijspeert and Kodjabachian [36],

FIG. 4. Phase-averaged velocity profiles at $x/c = 1.5$.

and Tytell and Lauder [37]). However, due to the complex interaction between the fluid and the deformable body of the animal, fundamental questions regarding thrust generation and its relation to the kinematics of the swimmer's body, the

efficiency, and viscous effects remain controversial. In particular, investigations of the undulatory propulsion mechanisms of anguilliform fish (e.g., *Anguilla anguilla*) started with the pioneering work of Gray [38]. In contrast to carangiform fish,

FIG. 5. Phase-averaged velocity profiles at $x/c = 2$.

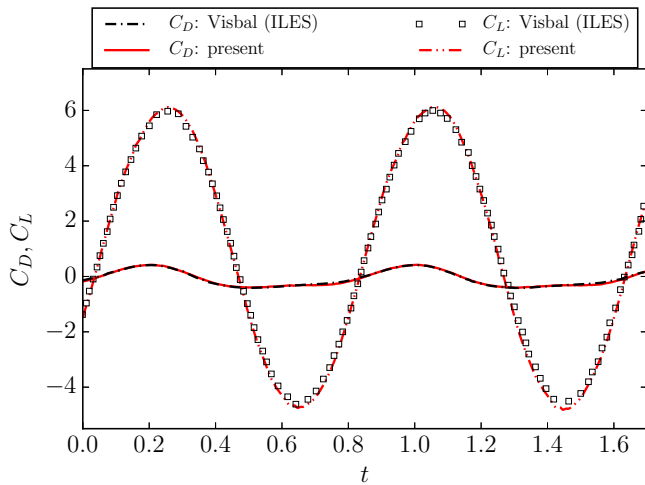


FIG. 6. Evolution of lift and drag coefficient over three exemplary cycles of a plunging airfoil.

anguilliform swimmers generate thrust by passing a transverse wave down their body, therefore utilizing, to a varying degree, the whole body for thrust generation and not just the tail. The hydrodynamics of aquatic locomotion for undulatory swimmers was studied for inviscid flow in the works of Lighthill [39–42], Wu [43], Hess and Videler [44], and Cheng *et al.* [45].

Simple algebraic models predict a reverse von Karman vortex street for optimized swimming performance, where the wake consists of a double row of single vortices when the ratio of the swimming speed and the body wave speed is less than 1 [40].

On the experimental side, PIV has proven itself as a valuable tool to quantify the flow field generated by the swimmer. The first visualization using two-dimensional PIV for freely swimming juvenile eel was reported by Müller *et al.* [46]. They found a linearly increasing flow velocity from head to tail, suggesting continuous thrust generation along the body. In the wake, they observed a double row of double vortices with little backward momentum, which is generated by a start-stop vortex shed from the tail and a separate vortex produced along the body, so-called protovortices, for each half tail-beat. They conjectured that the wake morphology is caused by a phase lag between the primary start-stop vortex and the body-generated circulation. Subsequent studies using high-resolution PIV were conducted by Tytell and Lauder [37]. Their results were, in general, similar to those of Müller *et al.* [46] but differences were noticed regarding the protovortices. Only a negligible phase lag with low vorticity was reported, resulting in a single, combined primary start-stop vortex per half tail-beat. The discrepancies were attributed to the lower PIV resolution and the seemingly accelerating eel in Müller *et al.* [46] compared to the steadily swimming fish in Tytell and Lauder [37]. In the work of Tytell and Lauder [37], they stated the following mechanism for the generation of the wake: A primary start-stop vortex is shed when the tail changes direction. Due to the acceleration of the tail from one side to the other, a low-pressure region develops in the posterior part of the body, drawing fluid in the lateral direction, which is shed off the tail and stretches the primary

vortex into an unstable shear layer which rolls up into two separate, corotating vortices, which they termed the secondary vortex. Thus, the primary vortex from one half tail beat and the secondary vortex from the subsequent comprise the boundaries of each lateral jet. Notable was the lack of any significant downstream flow for steady swimming as previously observed for carangiform fish and interpreted as thrust generation mechanism. This was further supported by the observation of only a slight upstream inclination of the lateral jets.

On the numerical front, only a few detailed studies may be found in the literature (see, e.g., Carling *et al.* [47], Kern and Koumoutsakos [3], Borazjani and Sotiropoulos [48], and Borazjani and Sotiropoulos [49]). The first two-dimensional viscous flow simulations of self-propelled anguilliform fish were reported in the work of Carling *et al.* [47]. In contrast to the experimentally observed wake morphology, these simulations indicated a single, large vortex ring wrapping around the eel, resulting in an upstream flow, where thrust is almost exclusively produced along the body and not the tail tip. Three-dimensional simulations were recently conducted by Kern and Koumoutsakos [3] using the finite-volume approach of the commercial software package STAR-CD with a first-order discretization in time and a second-order discretization in space. Apart from prescribing a reference motion of the fish, as proposed by Carling *et al.* [47], an evolutionary algorithm was employed to obtain the body motion as a result of optimizing for burst swimming speed and efficiency. Their results indicate that large-amplitude tail undulation in combination with a straight anterior body produce most of the thrust at the tail and are optimal kinematics in the burst swimming mode. On the other hand, optimal kinematics for efficient swimming were obtained for an undulation of the entire body, where thrust is generated with half the body and not just the tail. For all swimming patterns, the wake morphology did not differ qualitatively and is in agreement with the experimental observations of Tytell and Lauder [37], exhibiting a double row of single vortex rings with lateral jets.

Despite the valuable contributions mentioned above, further quantitative analysis is needed for conclusive results. The ease of data extraction, its analysis, and the precise control over the body's kinematics make numerical experiments useful for these studies. However, issues related to deforming geometries and the complex fluid-structure interaction are challenging for numerical solvers, thus explaining the sparsity of these simulations in the literature. The KBC model, on the other hand, is a highly efficient approach, which allows for a simple implementation of complex, deforming bodies using Cartesian meshes and is therefore ideally suited for this type of problem. The moving wall-boundary condition of Dorschner *et al.* [21] allows for non-body-fitted meshes, which significantly reduces and simplifies the process of grid generation. This, combined with the superior stability of KBC models compared to LBGK, the intrinsic parallelizability, and efficiency of LBM make these the best available tools for Direct Numerical Simulation (DNS) investigations of such complex flows. Therefore, as a first step to access the predictive capabilities of the proposed scheme, we conduct a simulation of a self-propelled anguilliform swimmer analogous to Kern and Koumoutsakos [3].

Geometrically, the body of the anguilliform swimmer with length L is modeled by spatially varying ellipsoidal cross

sections as in Kern and Koumoutsakos [3] for which the half axes $w(s)$ and $h(s)$ are defined as analytical functions of the arc-length s as

$$w(s) = \begin{cases} \sqrt{2w_h s - s^2} & 0 \leq s < s_b, \\ w_h - (w_h - w_t) \left(\frac{s-s_b}{s_t-s_b} \right)^2 & s_b \leq s < s_t, \\ w_t \frac{L-s}{L-s_t} & s_t \leq s \leq L, \end{cases} \quad (23)$$

where $w_h = s_b = 0.04L$, $s_t = 0.95L$, and $w_t = 0.01L$. For the height $h(s)$, an elliptical curve is prescribed as

$$h(s) = b \sqrt{1 - \left(\frac{s-a}{a} \right)^2}, \quad (24)$$

where the half axes are given as $a = 0.51L$ and $b = 0.08L$. As proposed by Carling *et al.* [47], the swimmer undergoes a prescribed time-dependent lateral deformation of the body's center line as

$$y(s, t) = 0.125\beta L \frac{s/L + 0.03125}{1.03125} \sin \left[2\pi \left(\frac{s}{L} - \frac{t}{T} \right) \right], \quad (25)$$

where T is the undulation period and β is an initial ramping function to assure a smooth transition from the initially resting body according to

$$\beta = \begin{cases} \frac{1 - \cos(\pi t/T)}{2} & 0 \leq t \leq T, \\ 1 & t > T. \end{cases} \quad (26)$$

The feedback from the fluid onto the body is prescribed by Newton's equations as

$$m_s \ddot{\mathbf{x}}_s = \mathbf{F}, \quad (27)$$

$$\frac{d\mathbf{I}_s \boldsymbol{\omega}_s}{dt} = \boldsymbol{\omega}_s \dot{\mathbf{I}}_s + \mathbf{I}_s \dot{\boldsymbol{\omega}}_s = \mathbf{M}, \quad (28)$$

where m_s , $\mathbf{I}_s$, $\mathbf{x}_s$, and $\boldsymbol{\omega}_s$ are the mass, the inertia tensor, the center of mass, and the angular velocity, respectively. The force and torque acting from the fluid onto the object are denoted by $\mathbf{F}$ and $\mathbf{M}$, respectively. As in the work of Kern and Koumoutsakos [3], the complexity of the system is reduced by only considering the torque corresponding to the yaw.

In the framework of the LB simulation, the geometrical model is represented as a triangulated surface composed of elliptical disks along the center line. The center line was decomposed into 150 segments, which has proven to be sufficient. This allows for a straightforward computation of all geometrical properties, including the time derivative of the inertia tensor. The velocity at the intersection points, needed for the boundary conditions, was computed based on a finite-difference scheme and a barycentric interpolation for the given time-dependent deformation. As above, the equations of motion are solved using an Euler integration scheme. Higher-order integration schemes have been tested but marginal differences were observed due to the relatively small time step used. Further, no smoothing or low-pass filtering of the hydrodynamic forces were applied, unlike in Kern and Koumoutsakos [3].

The simulations are carried out with a uniform grid using a domain of $[8L \times 4L \times L]$, the swimmer is resolved by $L = 200$ lattice points, and the undulation period is taken to be

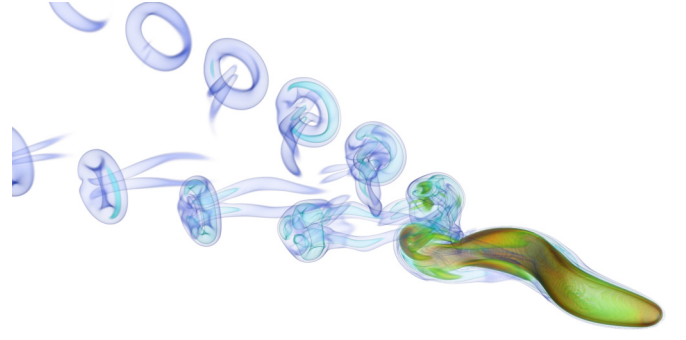


FIG. 7. Volume rendering of vorticity of the swimmers wake, showing the typical double row vortex street (see also the movie in the Supplemental Material [50]).

$T = 10^4$ lattice time steps. As in Kern and Koumoutsakos [3], the Reynolds number is taken as $\text{Re} = (L^2/T)/\nu = 7142$.

A volume rendering of vorticity shows the wake of the swimmer in Fig. 7. In agreement with Kern and Koumoutsakos [3], Tytell and Lauder [37], and Tytell [51], the wake consists of a double row of single vortices. Furthermore, we confirm the measurements from Tytell and Lauder [37] and observe a primary vortex being shed from the tail when it changes direction. The fluid drawn in the lateral direction by the accelerating tail and the development of two separate corotating vortices as a result of an unstable shear layer roll-up can also be seen in Fig. 7 and the Supplemental Material [50].

Quantitatively, we compare the evolution of the forward and the lateral velocity of the swimmer in Fig. 8. Overall, the agreement is good. Minor discrepancies are observed during acceleration but are attributed to different ramping functions used here, Eq. (26), and in Kern and Koumoutsakos [3]. The differences in the asymptotic forward and lateral velocity are within the range of expectation due to the different evolution algorithms. While Kern and Koumoutsakos [3] report an asymptotic forward velocity of $\bar{U}_{\parallel} = 0.4$ with an amplitude of 0.01, the present simulation yields $\bar{U}_{\parallel} = 0.415$ with an amplitude of 0.005. The lateral velocity $U_{\perp}$ has a zero mean for

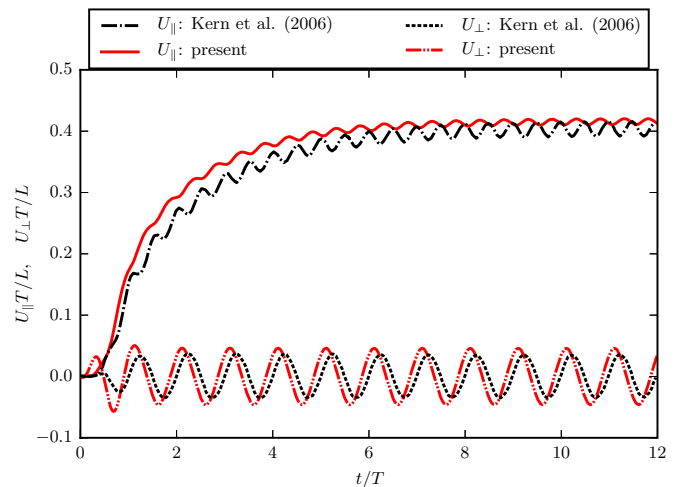


FIG. 8. Temporal evolution of the forward velocity $U_{\parallel}$ and lateral velocity $U_{\perp}$.

both the reference and the present simulation. An amplitude of 0.046 is measured in the present simulation, whereas Kern and Koumoutsakos [3] measure 0.03. All simulations were checked for grid independence.

V. CONCLUSION

In this contribution we have presented a thorough study of the entropic multirelaxation time model in combination with the Grad boundary condition for moving and deforming objects in three dimensions. After validation using the classical benchmark of a sedimenting sphere under gravity, the accuracy and robustness of the proposed method was accessed in the simulation of a plunging airfoil in the transitional regime. The comparison with literature revealed excellent agreement in all quantities including phase-averaged velocity profiles in the near wake region as well as the evolution of aerodynamic loads, important for realistic two-way coupled simulations. Finally, for deforming objects, the simulation of an anguilliform

swimmer was considered and validated by comparison to the numerical investigations of Kern and Koumoutsakos [3]. These simulations together with the previous studies (Bösch *et al.* [7], Chikatamarla and Karlin [52], Dorschner *et al.* [18]) establish the predictive capabilities of KBC models for complex, moving and deforming objects, where a simple grid convergence is sufficient to assert their validity. The above simulations of complex flows, when viewed together with the efficiency of KBC models and the ease of implementing boundary conditions, demonstrate that the present method may be a viable alternative for engineering-related fluid dynamics.

ACKNOWLEDGMENTS

This work was supported by the European Research Council (ERC) Advanced Grant No. 291094-ELBM and the ETH Grant No. ETH-32-14-2. The computational resources at the Swiss National Super Computing Center CSCS were provided under the Grants No. s492 and No. s630.

-
- [1] R. Howell, N. Qin, J. Edwards, and N. Durrani, *Renew. Energy* **35**, 412 (2010).
 - [2] M. Schmitt, C. E. Frouzakis, A. G. Tomboulides, Y. M. Wright, and K. Boulouchos, *Phys. Fluids* **26**, 035105 (2014).
 - [3] S. Kern and P. Koumoutsakos, *J. Exp. Biol.* **209**, 4841 (2006).
 - [4] M. D. De Tullio, A. Cristallo, E. Balaras, and R. Verzicco, *J. Fluid Mech.* **622**, 259 (2009).
 - [5] A. E. Randles, Modeling cardiovascular hemodynamics using the lattice boltzmann method on massively parallel supercomputers, Ph.D. thesis (2003).
 - [6] S. Ansumali, I. V. Karlin, C. E. Frouzakis, and K. B. Boulouchos, *Physica A* **359**, 289 (2006).
 - [7] F. Bösch, S. S. Chikatamarla, and I. V. Karlin, *Phys. Rev. E* **92**, 043309 (2015).
 - [8] N. Frapolli, S. S. Chikatamarla, and I. V. Karlin, *Phys. Rev. E* **90**, 043306 (2014).
 - [9] N. Frapolli, S. S. Chikatamarla, and I. V. Karlin, *Phys. Rev. E* **92**, 061301(R) (2015).
 - [10] A. Mazloomi M, S. S. Chikatamarla, and I. V. Karlin, *Phys. Rev. Lett.* **114**, 174502 (2015).
 - [11] M. Mendoza, B. M. Boghosian, H. J. Herrmann, and S. Succi, *Phys. Rev. Lett.* **105**, 014502 (2010).
 - [12] D. D'Humières, in *Rarefied Gas Dynamics: Theory and Simulations*, Progress in Astronautics and Aeronautics (American Institute of Aeronautics and Astronautics, Washington, DC, 1994), pp. 450–458.
 - [13] I. V. Karlin, A. Ferrante, and H. C. Öttinger, *Europhys. Lett.* **47**, 182 (1999).
 - [14] H. Chen, S. Kandasamy, S. Orszag, R. Shock, S. Succi, and V. Yakhot, *Science* **301**, 633 (2003).
 - [15] J. Latt and B. Chopard, *Math. Comput. Simul.* **72**, 165 (2006).
 - [16] M. Namburi, S. Krithivasan, and S. Ansumali, *Sci. Rep.* **6**, 27172 (2016).
 - [17] I. V. Karlin, F. Bösch, and S. S. Chikatamarla, *Phys. Rev. E* **90**, 031302(R) (2014).
 - [18] B. Dorschner, F. Bösch, S. Chikatamarla, K. Boulouchos, and I. V. Karlin, *J. Fluid Mech.* **801**, 623 (2016).
 - [19] C. S. Peskin, *J. Comput. Phys.* **25**, 220 (1977).
 - [20] K. Suzuki and T. Inamuro, *Comput. Fluids* **49**, 173 (2011).
 - [21] B. Dorschner, S. Chikatamarla, F. Bösch, and I. V. Karlin, *J. Comput. Phys.* **295**, 340 (2015).
 - [22] G. McGowan, A. Gopalathnam, M. Ol, J. Edwards, and D. Fredberg, in *46th AIAA Aerospace Sciences Meeting and Exhibit, Aerospace Sciences Meetings* (American Institute of Aeronautics and Astronautics, 2008).
 - [23] M. R. Visbal, *AIAA J.* **47**, 2685 (2009).
 - [24] S. Ansumali, I. V. Karlin, and H. C. Öttinger, *Europhys. Lett.* **63**, 798 (2003).
 - [25] A. N. Gorban and I. V. Karlin, *Invariant Manifolds for Physical and Chemical Kinetics*, Vol. 660, Lecture Notes in Physics (Springer, Berlin, Heidelberg, 2004).
 - [26] S. Tao, J. Hu, and Z. Guo, *Comput. Fluids* **133**, 1 (2016).
 - [27] T. A. Johnson and V. C. Patel, *J. Fluid Mech.* **378**, 19 (1999).
 - [28] G. Eitel-Amor, M. Meinke, and W. Schröder, *Comput. Fluids* **75**, 127 (2013).
 - [29] B. Wen, C. Zhang, and H. Fang, *Entropy* **17**, 8240 (2015).
 - [30] N. Li, H. Liu, and Y. Su, *PLoS ONE* **12**, e0174740 (2017).
 - [31] K. Ou, P. Castonguay, and A. Jameson, *49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition* (2011).
 - [32] L. G. Margolin and W. J. Rider, *Int. J. Numer. Methods Fluids* **39**, 821 (2002).
 - [33] L. G. Margolin, W. J. Rider, and F. F. Grinstein, *J. Turbul.* **7**, N15 (2006).
 - [34] B. Dorschner, N. Frapolli, S. S. Chikatamarla, and I. V. Karlin, *Phys. Rev. E* **94**, 053311 (2016).
 - [35] O. Ekeberg, *Biol. Cybern.* **69**, 363 (1993).
 - [36] A. J. Ijspeert and J. Kodjabachian, *Artif. Life* **5**, 247 (1999).
 - [37] E. D. Tytell and G. V. Lauder, *J. Exp. Biol.* **207**, 1825 (2004).
 - [38] J. Gray, *J. Exp. Biol.* **10**, 88 (1933).
 - [39] M. J. Lighthill, *J. Fluid Mech.* **9**, 305 (1960).
 - [40] M. J. Lighthill, *Annu. Rev. Fluid Mech.* **1**, 413 (1969).
 - [41] M. J. Lighthill, *J. Fluid Mech.* **44**, 265 (1970).
 - [42] M. J. Lighthill, *Proc. Roy. Soc. Lond. B: Biol. Sci.* **179**, 125 (1971).
 - [43] T. Y.-T. Wu, *Adv. Appl. Mech.* **11**, 1 (1971).

- [44] F. Hess and J. J. Videler, *J. Exp. Biol.* **109**, 229 (1984).
- [45] J.-Y. Cheng, L.-X. Zhuang, and B.-G. Tong, *J. Fluid Mech.* **232**, 341 (1991).
- [46] U. K. Müller, J. Smit, E. J. Stamhuis, and J. J. Videler, *J. Exp. Biol.* **204**, 2751 (2001).
- [47] J. Carling, T. L. Williams, and G. Bowtell, *J. Exp. Biol.* **201**, 3143 (1998).
- [48] I. Borazjani and F. Sotiropoulos, *J. Exp. Biol.* **211**, 1541 (2008).
- [49] I. Borazjani and F. Sotiropoulos, *J. Exp. Biol.* **212**, 576 (2009).
- [50] See Supplemental Material (movie) at <http://link.aps.org/supplemental/10.1103/PhysRevE.95.063306> showing volume rendering of vorticity for a self-propelled anguilliform swimmer.
- [51] E. D. Tytell, *J. Exp. Biol.* **207**, 3265 (2004).
- [52] S. Chikatamarla and I. V. Karlin, *Physica A* **392**, 1925 (2013).