ALE-LB for compressible flows

Arbitrary Lagrangian-Eulerian formulation of lattice Boltzmann model for compressible flows on unstructured moving meshes

M. H. Saadat\(^1\) and I. V. Karlin\(^{1,2}\)

Department of Mechanical and Process Engineering, ETH Zurich, 8092 Zurich, Switzerland

(Dated: 12 February 2020)

We propose the application of the arbitrary Lagrangian-Eulerian (ALE) technique to a compressible lattice Boltzmann model for the simulation of moving boundary problems on unstructured meshes. To that end, the kinetic equations are mapped from a moving physical domain into a fixed computational domain. The resulting equations in the computational domain are then numerically solved using the second-order accurate finite element reconstruction on an unstructured mesh. It is shown that the problem regarding the geometric conservation law (GCL), which needs a special treatment in the ALE Navier-Stokes solvers, does not appear here and the model satisfies the GCL exactly. The model is validated with sets of simulations including uniform flow preservation and compressible flow past airfoil with plunging and pitching motions at different Mach numbers. It is demonstrated that the results are in good agreement with the experimental and other available numerical results in the literature. Finally, in order to show the capability of the proposed solver in simulating high-speed flows, transonic flow over pitching airfoil is investigated. It is shown that the proposed model is able to capture the complex characteristics of this flow which involves multiple weak shock waves interacting with the boundary and shear layers.

I. INTRODUCTION

In recent years, there is a growing interest in studying both numerically\(^1\) and experimentally\(^2\) fluid flows in moving and deforming geometries in many physical phenomena and novel engineering applications. For example, in flapping flights of birds and insects, it is the motion of aerodynamic surfaces that produces thrust for forward motion and sustainable lift for airborne; or in marine animals such flapping motion generates propulsive and manoeuvring forces.\(^3\) Understanding the underlying aerodynamics of these phenomena provides researchers with valuable insight into the origin of flight and its subsequent evolution in different species.\(^4\) Moreover, these natural phenomena have been a rich source of inspiration in design of engineering devices such as robotic devices, micro-air vehicles or in novel turbines that extract energy form wind and tidal waves using flapping foil motion. The flapping foil turbine concept is promising in turbine technologies as it is expected to be more efficient than vertical- and horizontal-axis turbines.\(^5\) (for a review see Young, Lai, and Platzer\(^6\), Xiao and Zhu\(^7\)). Other applications of flows with moving geometries appear in fluid-solid interactions (FSI) and rotor-stator flows to name a few.\(^8\)

Development of accurate and efficient numerical schemes for the simulation of fluid flows in complex domains remains a highly active research area in computational fluid dynamics (CFD). Presence of moving/deforming geometries in a flow adds another level of complexity to the computations, as it requires the numerical scheme not only to be able to handle moving domains, but to maintain accuracy and efficiency.\(^9\) From a physical point of view, moving domain problems usually involve vortex dominated unsteady flow with turbulence, separation and reattachment of the boundary layer. Numerically, capturing such a complex physics requires the numerical scheme to be accurate with small numerical dissipation.

The Lattice Boltzmann method (LBM) is a kinetic theory approach to CFD which has been proven as an accurate and reliable tool for the simulation of complex fluid flows ranging from turbulence\(^13\) and multiphase\(^14\) to micro-scale flows\(^15\) and compressible flow\(^16\) (for a review see Aidun and Clausen\(^17\)). In the LBM, populations \(f_i(x,t)\) associated with a set of discrete velocities \(\mathcal{C} = \{c_i, i = 0, \ldots, Q - 1\}\) are designed to recover the target equations of continuum mechanics in the hydrodynamic limit. The evolution of populations is based on simple rules of propagation along the discrete velocities \(\mathcal{C}\), and relaxation to a local equilibrium. This makes the LBM a simple and efficient alternative for conventional CFD solvers and an attractive candidate for simulation of flows with moving geometries.

For handling moving complex geometries, most of the existing LB realizations employ a fixed background regular Cartesian grid which cuts the immersed moving object. Imposing no-slip boundary condition on the moving object is then achieved either through adding a force term into the equations\(^12,13,14\) or by replacing missing populations with some suitable approximation like non-equilibrium extrapolation\(^15\) or Grad’s approximation\(^16\).

Another approach which has widely been used in the Navier-Stokes (NS) framework for simulating moving domain problems is based on the so-called arbitrary Lagrangian-Eulerian (ALE) method\(^17,21\). In this method, the governing equations in the physical domain which is moving in space and time, are mapped into a fixed computational domain and then the resulting transformed equations are solved numerically\(^18\). ALE method thus gives flexibility in handling moving domain problems as the physical mesh can move with arbitrary velocity independent of the fluid velocity\(^20\). Another advantage of the ALE method is that it can handle moving/deforming domains with body-fitted mesh which is of crucial importance for high-Reynolds number flow simulations where small scale structures need to be resolved accurately.

\(^{a1}\)Electronic mail: karlin@lav.mavt.ethz.ch
For problems involving rigid motion of objects and also problems with small deformation, it is possible to derive an analytical formula for the mapping function between physical and computational domains and that, in turn, greatly simplifies the computations. However, in problems with very large deformation, mapping function can become highly nontrivial and re-meshing might be required, which is computationally expensive and can easily make the simulation unfeasible.

While numerous studies have been done on the ALE-NS solvers, limited number of works can be found in the literature about applying ALE method in the context of LBM. Noteworthy is the model proposed by Meldi, Vergnault, and Sagaut which is based on the combination of ALE and over-set grid methods and was used for simulation of low-speed incompressible flow.

The aim of this paper is to investigate the application of the ALE method in LB framework for the simulation of moving domain problems. It is well known that most of the conventional LB models in the literature are limited to low-speed isothermal incompressible flows. That is due to the insufficient isotropy and lack of Galilean invariance of the standard lattices (D2Q9 in two dimensions and D3Q27 in three dimensions, where \( DdQn \) model refers to \( d \) dimension model with \( n \) discrete velocities). A systematic way to overcome this severe limitation could be achieved by increasing the number of discrete velocities and use the hierarchy of higher-order (multi-speed) lattices. However, employing high-order lattices comes at the price of increasing the computational cost. Another recent approach, which maintains the simplicity and efficiency of the standard lattices, is to introduce appropriate correction terms into the kinetic equations in order to compensate the error terms resulting from the low symmetry of the standard lattices (see e.g. the models proposed by Prasianakis and Karlin\( ^{22} \), Feng, Sagaut, and Tao\( ^{23} \) and Huang, Wu, and Adams\( ^{24} \). Following this approach, we have recently introduced a compressible LB model on standard lattices which can recover the full Navier-Stokes-Fourier (NSF) equations with adjustable Prandtl number and adiabatic exponent in the hydrodynamic limit\( ^{22} \). We then enriched our model by employing the concept of the shifted lattice\( ^{25} \) and showed that the model works pretty well for compressible flows up to moderate supersonic regime with shock waves.

In this paper, we apply the ALE method to the compressible LB model\( ^{27} \) as it gives us a unified flow solver which covers subsonic to moderately supersonic regimes. To the best of our knowledge, LBM has not been investigated for the simulation of compressible flows with moving bodies. It should, however, be emphasized that the ALE formulation given below is general and, in principle, can be applied to any lattice kinetic model, including incompressible Mattila, Philippi, and Hegele Jr\( ^{29} \), Bösch, Chikatamarla, and Karlin\( ^{30} \), thermal Karlin, Sichau, and Chikatamarla\( ^{31} \) or compressible models Frapolli, Chikatamarla, and Karlin\( ^{32} \).

Since transformed equations should be solved in the ALE method, exact propagation on space-filling lattice is not possible anymore and an off-lattice scheme should be adopted. Among various off-LB schemes existing in the literature, such as finite difference (FD) or finite volume (FV) LB schemes\( ^{33} \), the semi-Lagrangian scheme based on the finite-element interpolation\( ^{33} \) has been shown to be an efficient and accurate scheme which maintains the advantage of high Courant–Friedrichs–Lewy (CFL) number, while removes the restriction of using regular lattice. The semi-Lagrangian finite element scheme makes it also possible to employ unstructured body-fitted grid which gives us more flexibility in handling complex geometries and is more efficient in flows with high-Reynolds number. The application of the semi-Lagrangian scheme with finite element interpolation has been studied for incompressible LB models in both laminar\( ^{35} \) and turbulent regimes\( ^{36} \).

The rest of the paper is organized as follows: The ALE formulation of the LB model is discussed in Sec.II. The detailed numerical implementation of the ALE-LB model on unstructured mesh is presented in Sec.III. It is also shown that the ALE-LB model satisfies the geometric conservation law (GCL) exactly. The kinetic equations of the compressible LB model along with the implementation of no-slip wall boundary conditions are briefly reviewed in Sec.IV. In Sec.V, the model is validated through simulation of benchmark test-cases, including free-stream preservation and compressible flow over NACA0012 airfoil in plunging and pitching motions. Moreover, in order to test the model’s performance in simulating high-speed flows, the transonic flow over pitching airfoil is considered in this section. Finally, some conclusions are drawn in Sec.VI.

II. ALE FORMULATION OF THE LATTICE BOLTZMANN

Consider the Boltzmann equation in a physical domain \((x,t)\)

\[
\frac{\partial f_i}{\partial t} + c_i \nabla_x f_i = \Omega_i, \tag{1}
\]

where \(f_i(x,t)\) are populations of discrete velocities \(c_i, i = 0, \ldots, Q - 1\), and \(\Omega_i\) is the collision operator. The goal here is to transform Eq. (1) from a physical domain \((x,t)\) to a fixed computational domain \((X, t_0)\).

We assume that there exists a continuous time dependent mapping between physical and computational domains, denoted by \(G\), such that \(x = G(X, t)\). The time derivative in Eq. (1) can then be re-written as

\[
\frac{\partial f_i}{\partial t} = \frac{d f_i}{d t} - V_G \cdot \nabla_X f_i = \frac{d f_i}{d t} |_X - V_G \cdot \nabla_X f_i, \tag{2}
\]

where the time derivative \(\frac{d f_i}{d t} |_X\) is at constant \(X\), spatial derivatives are taken with respect to \(X\), and \(V_G\) denotes the mapping velocity as

\[
V_G = \frac{d G}{d t} |_X . \tag{3}
\]

Using a simple chain rule, the spatial terms in Eq. (1) can also be written as

\[
\nabla_x f_i = g^{-1} \nabla_X f_i. \tag{4}
\]
Here, \( g^{-1} \) is the inverse of the Jacobian matrix of mapping, which for two-dimensional problems can be computed as

\[
g^{-1} = \frac{1}{\det g} \begin{bmatrix} y y & -y x \\ -x y & x x \end{bmatrix}, \tag{5}\]

where \( x_x, x_y, y_x, \) and \( y_y \) are mapping metrics and

\[
\det g = x_x y_y - x_y y_x, \tag{6}\]

is the determinant of the Jacobian matrix. Substituting Eq. (2) and Eq. (4) into Eq. (1)

\[
\frac{\partial f_i}{\partial t} \bigg|_{X + c_i g^{-1} \nabla_X f_i - V_G \cdot \nabla_X f_i} = \Omega_i, \tag{7}\]

which can be further simplified as

\[
\frac{\partial f_i}{\partial t} \bigg|_{X + \left( g^{-1 \cdot T} \right) c_i - V_G} \cdot \nabla_X f_i = \Omega_i, \tag{8}\]

where the superscript \( T \) denotes the transpose of a matrix. By defining transformed discrete velocities \( \hat{c}_i \) as

\[
\hat{c}_i = \left( g^{-1 \cdot T} \right) c_i - V_G, \tag{9}\]

the Boltzmann equation in a fixed computational domain can be written in a simple form as

\[
\frac{\partial f_i}{\partial t} \bigg|_{X + \hat{c}_i \cdot \nabla_X f_i} = \Omega_i. \tag{10}\]

As it can be seen, the only difference between Eq. (10) and Eq. (1) is in discrete velocities. We can therefore conclude that, the ALE method is applicable to any lattice kinetic model just by using the transformed discrete velocities as defined in Eq. (9).

Now, Eq. (10) can be discretized using conventional scheme used in the standard LB, i.e. through propagation and collision steps

\[
f_i(X, t) - f_i(X + \hat{c}_i \delta t, t - \delta t) = \sum_{s=1}^{9} N_s(\xi_{dp}) f_i(\xi_s, t - \delta t), \tag{12}\]

where \( N_s(\xi_{dp}) \) denote the values of the shape functions, written in the local coordinate system \( \xi = (\xi, \eta), (-1 \leq \xi, \eta \leq 1) \), at the departure point (red square in Fig. 1). \( f_i(\xi_s, t - \delta t) \) are the values of populations \( f_i \) at the collocation nodes (red circles in Fig. 1), and \( s = 9 \) is number of collocation points.

Therefore, semi-Lagrangian propagation on unstructured finite-element mesh requires two steps: First, computing the local coordinates of the departure point \( \xi_{dp} \) (see Fig. 1). Here, a bi-linear transformation is used to transform the computational cells into a reference unit cell. Thus, finding the local coordinates requires solving a non-linear system of equations resulting from

\[
X_{dp} = \sum_{s=1}^{4} N_s(\xi_{dp}) X_s. \tag{13}\]

Unlike stationary case, where the location of departure point for each node is constant during the simulation, in moving case, the departure point is moving and therefore, the non-linear equation (13) should be solved in each time step. Second, computing the values of the populations at the departure point by means of the values of the populations at collocation nodes (red circles), i.e. by using Eq (12).

After propagation, the post-collision populations are computed.
A. Geometric conservation law

The problem of geometric conservation law (GCL) was first introduced in Thomas and Lombard\cite{35}, where it was shown that the numerical discretization errors associated with mapping metrics can induce errors in the computed flow field which might lead to numerical instabilities\cite{36}. This problem has been widely studied in the NS solvers and different strategies have been proposed for satisfying the GCL in that context\cite{35,36,37,38}.

In order to mathematically check the GCL, a uniform flow should satisfy Eq. (11). As the collision term vanishes with constant uniform flow, we just need to insert a constant solution \( f_i(X, t) = f_i^0 \) into Eq. (12)

\[
f_i^0 = \sum_{s=1}^{N_s} N_s f_i^0,
\]

and, therefore, the present model satisfies the GCL exactly.

Before proceeding to results section, we briefly describe the compressible LB model used in this study.

IV. KINETIC EQUATIONS

The kinetic model used in this study is the two-population compressible LB model on standard lattices, which recovers the full NSF equations in the hydrodynamic limit\cite{35}.

The kinetic equations of this model written in the computational domain are as follows

\[
\begin{align*}
    f_i(X, t) - f_i(X - \hat{c}_i \delta t, t - \delta t) &= \alpha (f_i^{eq} - f_i) + \delta t \phi_i, \\
g_i(X, t) - g_i(X - \hat{c}_i \delta t, t - \delta t) &= \omega (g_i^{eq} - g_i) + \omega_1 (g_i^{eq} - g_i),
\end{align*}
\]

where \( \hat{c}_i \) are transformed discrete velocities computed using Eq. (9)

\[
\hat{c}_i = (g^{-1} \cdot c_i - V_G),
\]

and standard set of discrete velocities \( c_i \) in two dimensions and with \( Q = 9 \) (D2Q9 model) is defined as

\[
c_i = (c_{ix}, c_{iy})^T, \quad i = 0, ..., Q - 1; \quad c_{i\alpha} \in \{-1, 0, +1\}, \quad \alpha = x, y.
\]

The \( \phi_i \) terms in Eq. (16) are correction terms responsible for canceling out the spurious terms in the momentum equation, resulting from low symmetry of the standard lattices (for details of deriving correction terms see Saadat, Bösch, and Karlin\cite{39}). \( g_i^{eq} \) is a quasi-equilibrium population, and \( f_i^{eq}, g_i^{eq} \) are local equilibria which satisfy the local conservation laws for the density \( \rho \), momentum \( \rho u \) and total energy \( \rho E \),

\[
\begin{align*}
    \sum_{i=0}^{Q-1} \{1, c_i\} f_i &= \sum_{i=0}^{Q-1} \{1, c_i\} f_i^{eq} = \{\rho, \rho u\}, \\
    \sum_{i=0}^{Q-1} g_i &= \sum_{i=0}^{Q-1} g_i^{eq} = 2 \rho E.
\end{align*}
\]

The temperature is defined by

\[
T = (1/C_v)(E - u^2/2),
\]

Below, a system of units is used where the universal gas constant is set to one, \( R = 1 \). Consequently, \( C_p = C_v + 1 \) is the specific heat at constant pressure and the Prandtl number is \( Pr = C_p\mu/\kappa; \gamma = C_p/C_v \) is the adiabatic exponent which can be freely adjusted.

The equilibrium populations can be written in a product-form as

\[
f_i^{eq} = \rho \Phi_{\alpha} \Phi_{\beta},
\]

where

\[
\begin{align*}
    \Phi_{-1} &= -u_\alpha + u_{\alpha}^2 + T, \\
    \Phi_{0} &= 1 - (u_\alpha^2 + T), \\
    \Phi_{+1} &= u_\alpha + u_{\alpha}^2 + T,
\end{align*}
\]

The populations \( g_i^{eq}, g_i^* \) are constructed using the following general form

\[
G_i = W_i \left( M_{00} + \frac{M_{\alpha\alpha} c_\alpha}{T} + \frac{(M_{\alpha\beta} - M_{\alpha\alpha} T \delta_{\alpha\beta}) (c_\alpha c_\beta - T \delta_{\alpha\beta})}{2T^2} \right) + \Psi_i,
\]

where \( W_i \) are temperature-dependent weights

\[
W_i = W_{c\alpha} W_{c\beta},
\]

with

\[
\begin{align*}
    W_{-1} &= T \frac{2}{T}, \\
    W_0 &= 1 - T, \\
    W_{+1} &= T \frac{2}{T},
\end{align*}
\]

and other terms required for the computations are provided in TableII and defined as
TABLE I. Moments needed for the computation of $g_i^{eq}$ and $g_i^t$.

| $G_i$ | $M_0$ | $M_α$ | $M_{αβ}$ | $Ψ_i$ |
|-----|-----|-----|-----|-----|
| $s_i^{eq}$ | $2pE$ | $q_i^{eq}$ | $R_{αβ}$ | $ψ_i$ |
| $g_i^t$ | $2pE$ | $q_i^t$ | $R_{αβ}$ | $ψ_i$ |

$q_i^{eq} = \sum_{i=0}^{Q-1} c_{ia} s_i^{eq} = 2ρu_α(E + T)$, \hspace{1cm} (34)

$R_{αβ}^{eq} = \sum_{i=0}^{Q-1} c_{ia} c_{ib} s_i^{eq} = 2ρE(T δ_{αβ} + u_αu_β)$ 

$+ 2pT(T δ_{αβ} + 2u_αu_β)$, \hspace{1cm} (35)

$q_i^t = \sum_{i=0}^{Q-1} c_{ia} s_i^t = 2u_β(p_{αβ} - p_{αβ}^{eq})$, \hspace{1cm} (36)

$P_{αβ}^{eq} = \sum_{i=0}^{Q-1} c_{ia} c_{ib} f_i^{eq} = ρu_αu_β + ρT δ_{αβ}$, \hspace{1cm} (37)

$ψ_i = B_{αβ} \left( ρ (1 - 3T) \left( T^2 + 2u_α^2T + E u_α^2 \right) / T \right)$, \hspace{1cm} (38)

$B_{αβ} = \begin{cases} 1, \text{for } c_1 = 0, \\
- | c_α - 1/2 c_{αa}^2 |, \text{otherwise.} \end{cases}$

Note that, summation convention is used in above equations.

Finally, the correction terms $φ_i$ can be computed as

$φ_i = A_{αa}X_α$, \hspace{1cm} (39)

where

$X_α = - ρ \mu / (ρT) \partial_α Q_{αβγ}^t$, \hspace{1cm} (40)

$A_{αa} = c_α - 1/2 c_{αa}^2$, \hspace{1cm} (41)

and $Q_{αβγ}^t$ is the deviation of the the third-order equilibrium moment from the continuous Maxwell-Boltzmann moment (for further detail see Saadat, Bösch, and Karlin).

It is important to note that, spatial derivatives in the correction terms $φ_i$ (Eq. (39)) should also be transformed from the physical to computational domain. This can be done using a simple chain rule similar to Eq. (4). For a generic variable $K$, we can write

$\partial_x K = g^{-1} \partial_X K$. \hspace{1cm} (42)

Here, $g^{-1}$ is computed using Eq. (4), and

$\partial_X K = J^{-1} \sum s K_s \partial_x N_s$, \hspace{1cm} (43)

where $J^{-1}$ is the inverse of the Jacobian matrix of transformation of a computational cell to a unit cell computed with

$J^{-1} = \frac{1}{\text{det } J} \begin{bmatrix} \partial_η \gamma & - \partial_ξ \gamma \\ - \partial_η \chi & \partial_ξ \chi \end{bmatrix}$, \hspace{1cm} (44)

Note that, the nodes on the element edges are assigned to the element with the larger area.

A. No-slip wall boundary condition

The no-slip wall boundary condition (BC) used in this work is based on those proposed in Dorschner et al. The general idea is to replace the missing populations during the propagation step with the following expression

$f_i^{eq} = f_i^{eq} (ρ_{gt}, u_{tgt}, T_{tgt}) + δf_i^{t(1)} (ρ_{gt}, u_{tgt}, T_{tgt}, \nabla u_{tgt}, \nabla T_{tgt})$, \hspace{1cm} (45)

$s_i^{eq} = s_i^{eq} (ρ_{gt}, u_{tgt}, T_{tgt}) + δs_i^{t(1)} (ρ_{gt}, u_{tgt}, T_{tgt}, \nabla u_{tgt}, \nabla T_{tgt})$, \hspace{1cm} (46)

where $f_i^{eq}$, $g_i^{eq}$ are equilibrium parts computed from and $f_i^{t(1)}$, $g_i^{t(1)}$ are non-equilibrium parts and $ρ_{gt}$, $u_{tgt}$ and $T_{tgt}$ are target values which will be specified later. The non-equilibrium parts are obtained based on the Grad’s approximation and using the general formula (29) with the non-equilibrium moments given in Table [11].

$P_{αβ}^{t(1)} = - \frac{1}{c_v} \rho T \left( S_{αβ} - \frac{1}{c_v} \partial_α u_β δ_{αβ} \right)$, \hspace{1cm} (47)

$Q_{iα}^{t(1)} = - \frac{2}{c_v} \rho C_{p} T \partial_α T + 2u_β P_{αβ}^{t(1)}$, \hspace{1cm} (48)

$R_{αβ}^{t(1)} = - \frac{2}{c_v} \rho T \left[ S_{αβ} (E + 2T) + u_α \partial_β E + u_β \partial_α E \right]$, \hspace{1cm} (49)

where the strain rate tensor is

$S_{αβ} = \partial_α u_β + \partial_β u_α$. \hspace{1cm} (50)

For computing target values, if missing populations belong to points on the wall (black circles in Fig. 2), target veloci- ties are wall velocities, $u_{tgt} = u_{wall}$ and target density and temperature (for adiabatic wall) are obtained by setting

$\frac{∂ρ}{∂n}|_{wall} = 0$, \hspace{1cm} (51)

$\frac{∂T}{∂n}|_{wall} = 0$. \hspace{1cm} (52)

TABLE II. Moments needed for the computation of $f_i^{t(1)}$ and $g_i^{t(1)}$.

| $G_i$ | $M_0$ | $M_α$ | $M_{αβ}$ | $Ψ_i$ |
|-----|-----|-----|-----|-----|
| $f_i^{t(1)}$ | 0 | 0 | $P_{αβ}^{t(1)}$ | 0 |
| $g_i^{t(1)}$ | 0 | $Q_{iα}^{t(1)}$ | $R_{αβ}^{t(1)}$ | 0 |
\[ A \]

the distance from

Given the normal direction \( n \), its end point \( B \) and considering

distance from \( A \) to \( B \) as \( \| n \| = \delta t \), the values of density and
temperature at \( B \) can be evaluated using a finite element interpolation

\[
\rho_B = \sum_{s=1}^{9} N_s \rho_s,
\]

(55)

where \( N_s \) are shape functions and \( \rho_s \) are the magnitude of
density at nine collocation points (circles in Fig. 2). Once \( \rho_B \) is
found, the first-order approximation for the normal derivative is assumed

\[
\frac{\partial \rho}{\partial n} \bigg|_{wall} = \frac{\rho_B - \rho_A}{\| n \|} = 0.
\]

(56)

Therefore, the target value can be approximated as

\[
\rho_{tgt} = \rho_A = \rho_B.
\]

The same procedure is applied for computing target temperature \( T_{tgt} \). Note that if missing populations belong to points which do not lie on the wall boundaries (red circles in Fig. 2), the local quantities of the previous time step are used as target values.

The evaluation of spatial gradients in non-equilibrium parts is performed using (57). It was demonstrated in Dorschner et al.\cite{28} that the first-order accurate evaluation of spatial derivatives is sufficient.

V. RESULTS

In this section, the model presented above is tested numerically through simulation of benchmark cases for moving boundary problems. First, the GCL of the model is validated. Then, we investigate the flapping airfoil under pure plunging

and pitching motions, which is relevant in many physical applications including the flight of small fliers or micro air vehicles. All simulations are performed with \( \gamma = 1.4 \), \( \text{Pr} = 0.71 \), D2Q9 lattice model and the adiabatic wall assumption.

A. Free-stream preservation

The first test-case is to check the GCL of the model, i.e. to ensure the exact conservation of the free-stream condition under arbitrary movement of the mesh. We consider a uniform flow with \( Ma = u_{\infty}/\sqrt{T} = 0.2 \) and \( T = 0.2 \) in a square domain of size \( L = 8000 \). The mesh motion is defined through the following mapping function

\[
x(t) = X + 500 \sin(2\pi X/L) \sin((2\pi Y/L)\sin((2\pi t/\tau) ));
\]

(58)

\[
y(t) = Y + 500 \sin(2\pi X/L) \sin((2\pi Y/L)\sin((2\pi t/\tau) ));
\]

(59)

with the reference time \( \tau_0 = 1.5L/u_{\infty}. \) Figure 3 shows the mesh at two different non-dimensional times \( \tau^* = t u_{\infty}/L \). We compute the solution until non-dimensional time \( \tau^* = 1 \) using three different uniform grids, and the relative errors \( \varepsilon = \sum |u-u_{\infty}| / \sum |u_{\infty}| \) of the velocity \( u \) are shown in Table III. As it can be seen, the errors are found to be very small for different grids which demonstrate that the GCL is satisfied in the present model.

B. Flow over NACA0012 airfoil in plunging motion

We consider a flow over an airfoil in a plunging motion to test the capability of the solver in handling complex vortex-dominated flows. It is known that the flow over plunging airfoil produces thrust over a wide range of oscillation frequencies\cite{49}, the phenomenon known as Knoller–Betz effect\cite{50}. Another interesting phenomenon that has been observed in experiments\cite{50} is the formation of asymmetric deflected wake pattern at high Strouhal numbers, even in symmetrically plunging motions. Apart from experimental studies\cite{49,50}, several numerical studies\cite{11, 51, 52, 53} have also been done for understanding the physics behind the plunging airfoil and the mechanism of thrust generation.

Here, in order to take into account the effect of compressibility, the numerical setup is identical to the numerical study by Liang et al.\cite{11} based on the high-order accurate spectral difference ALE solution of the compressible NS equations (SDNS). A NACA0012 airfoil with chord length \( c = 200 \) is placed

TABLE III. Relative error \( \varepsilon \) of the velocity \( u \) for the free-stream

| Mesh(\Delta x/L) | \( \varepsilon \) |
|-----------------|-------------|
| 0.1             | 1.117 \times 10^{-15} |
| 0.05            | 4.028 \times 10^{-16} |
| 0.025           | 3.872 \times 10^{-16} |
in the center of a domain with the size \([40c \times 40c]\). The airfoil is undergoing a sinusoidal plunging motion prescribed as

\[
\begin{align*}
x(t) &= X, \\
y(t) &= Y - h \sin(\omega t),
\end{align*}
\]

(60)

where \(h\) and \(\omega\) are plunge motion amplitude and frequency, respectively. The Strouhal number is defined as

\[
Sr = h\omega / u_{\infty},
\]

(62)

and \(u_{\infty}\) is the free-stream velocity.

Two different scenarios are considered here: slow plunging and fast plunging motions. For both cases the Reynolds number based on the free-stream velocity \(u_{\infty}\) is set to \(Re = \rho_{\infty} u_{\infty} c / \mu = 1850\), the Mach number is \(Ma = u_{\infty} / \sqrt{T_{\infty}} = 0.2\) and the free-stream temperature is \(T_{\infty} = 0.3\).

1. **Slow plunging motion**

In the slow plunging motion, the plunge amplitude is \(h = 0.08c\) and the Strouhal number is \(Sr = 0.46\). We first compute this case using two different meshes with minimum cell sizes of \(\delta \approx 0.7\) (Mesh-1) and \(\delta \approx 0.5\) (Mesh-2), in order to investigate grid independence. Part of the mesh is shown in Fig. 4, where orthogonal grid is used close to the wall to accurately resolve the boundary layer and anisotropic unstructured grid is used elsewhere. In order to correctly capture the vortical patterns in the wake area, a high resolution mesh with cell size of \(\delta \approx 10\) is used in the rectangular domain around the airfoil. Moreover, to minimize the computational cost, the mesh outside of the rectangular domain is highly coarse which makes the ratio between largest and smallest cell size to be of approximately 1000.

The time evolution of the aerodynamic forces predicted by both grids are compared in Fig. 5. The lift coefficient is defined as \(c_l = F_l / (0.5 \rho_{\infty} u_{\infty}^2 c)\), where \(F_l\) is the total lift force acting on the airfoil and the drag coefficient is given by \(c_d = F_d / (0.5 \rho_{\infty} u_{\infty}^2 c)\), where \(F_d\) denotes the total drag force. As it can be seen in Fig. 5 lift coefficient varies symmetrically about zero mean, however, drag coefficient oscillates around a negative average value, which means that a small thrust is generated in this case. Moreover, the two grids give almost identical results which shows convergence to a grid independence solution. To validate the solver, the numerical results of Liang et al. over some cycles are also shown in Fig. 5. It is observed that the results are in good agreement.

Figure 6 shows the vorticity contours obtained by the present model in comparison with the experimental results reported by Jones, Dohring, and Platzer. Due to relatively low Strouhal number in this case, the leading and trailing edges separation results in an almost symmetric flow pattern which is very similar to the experiment.

2. **Fast plunging motion**

Next we consider the fast plunging motion of the NACA0012 airfoil which corresponds to a motion with \(h = 0.12c\) and \(Sr = 1.5\). The computation is performed using the mesh with the minimum cell size of \(\delta \approx 0.5\).

Figure 7 compares the vorticity contour obtained from the present model with the experimental results of Jones, Dohring, and Platzer. It can be seen that in this case, the spatial symmetry of the wake vortex pattern is lost and a deflected vortex street is generated. The deflected vortex pattern is travelling upward, because, according to Eq. 61, the first stroke is downward. It is also shown that the present model is able to capture dual-mode vortex street, in close resemblance with the experiment. The formation of dynamic stall vortex (DSV) near the leading edge of the airfoil is also observed.
in this figure. Dynamic stall vortices convect towards the trailing edge of the airfoil.

In order to validate the results quantitatively, we compare the time history of the aerodynamic forces computed over several periods by the present model with that of the SD-NS solver, excellent agreement is observed. As it is shown in Fig. 5, the maximum value of lift is larger than the slow plunging case and it oscillates symmetrically around a small mean value of about 1.43. The drag coefficient, on the other hand, is asymmetric and mainly negative which results in a net mean thrust.

C. Flow over NACA0012 airfoil in pitching motion

Now, we turn our attention to a flow over pitching airfoil. The experimental works conducted by Koochesfahani, Bohl and Koochesfahani and Mackowski and Williamson are among most comprehensive studies on flow over pitching airfoil in the incompressible regime, where they studied the vortical patterns in the wake and measured the thrust coefficient as the function of the reduced frequency. Experiments show that the thrust coefficient increases monotonically with pitching frequency. However, the pure pitching motion is not, in general, an effective mechanism for producing thrust. There are also several numerical studies in the literature about differ-
FIG. 6. Vorticity computed by the present model (top) and the experimental results reported by Jones, Dohring, and Platzer [40] (bottom), for slow plunging motion of NACA0012 airfoil with $h = 0.08c$ and $Sr = 0.46$. Contour levels are bounded between $-6 \leq \Omega c/u_{\infty} \leq 6$.

FIG. 7. Vorticity computed by the present model (top) and the experimental results reported by Jones, Dohring, and Platzer [40] (bottom), for fast plunging motion of NACA0012 airfoil with $h = 0.12c$ and $Sr = 1.5$. Contour levels are bounded between $-6 \leq \Omega c/u_{\infty} \leq 6$.

FIG. 8. Time evolution of lift (left) and drag (right) coefficients for fast plunging motion of NACA0012 airfoil with $h = 0.12c$, $Sr = 1.5$ and $Ma = 0.2$.

$$x(t) = (X - X_c)c(\theta) - (Y - Y_c)sin(\theta),$$  \hspace{1cm} (63)  
$$y(t) = (X - X_c)c(\theta) + (Y - Y_c)c(\theta),$$  \hspace{1cm} (64)

where $(X_c,Y_c)$ is the center of rotation, $\theta = Asin(\omega t)$ is the pitching angle, $A$ denotes the pitch motion amplitude and $\omega$ is the pitching frequency. The reduced frequency of pitching is defined as

$$k = \omega c/2 u_{\infty}. \hspace{1cm} (65)$$

First, the Mach number is considered to be $Ma = 0.08$ to avoid significant effect of compressibility and to compare the
FIG. 9. Vorticity computed by the present model (top) and the experimental results reported by Koochesfahani\textsuperscript{46} (bottom), for pitching motion of NACA0012 airfoil with $A = 2^\circ$ and $k = 6.68$. Contour levels are bounded between $-18 \leq \Omega/\omega_{\infty} \leq 18$.

results with the water tunnel experiment data\textsuperscript{2,47}. The simulation is performed at pitching amplitude of $A = 2^\circ$, reduced frequencies of $k = 0$ (stationary), $k = 6.68$ and $k = 10$ and at Reynolds number $Re = 12000$. The high Reynolds number makes this test-case more challenging, although the flow is still considered to be laminar. The mesh used for the computations has minimum cell size of $\delta \approx 0.2$ close to the wall.

The vortical pattern obtained by the present model is shown in Fig. 9 in comparison with the experimental results of Koochesfahani\textsuperscript{46}, where similar pattern can be observed. The vortex pattern of the present simulation are also quite consistent with other numerical results in the literature\textsuperscript{39,43}.

The time history of aerodynamic forces are presented in Figs. 10 and 11 for reduced frequencies of $k = 6.68$ and $k = 10$, respectively. In both cases, the lift force acting on the airfoil is only due to pressure term ($c_{l-p}$) while the contribution from the viscous force ($c_{l-v}$) vanishes. Under this condition, the average lift is zero. The drag force however, has contributions from both the pressure ($c_{d-p}$) and the viscous forces ($c_{d-v}$). There is an average drag force acting on the airfoil in the case of $k = 6.68$ and a small thrust in the case of $k = 10$.

To investigate the effect of compressibility and in accordance with the numerical simulation of Young and S. Lai\textsuperscript{48} based on the finite-difference discretization of compressible NSF equations, we repeated the simulations at higher Mach number $Ma = 0.2$. The time histories of lift and drag coefficients in this case at reduced frequency of $k = 10$ are shown in Fig. 11 in comparison with the results of the low Mach number case. It can be seen that the compressibility effect significantly changes the distribution of pressure force, while the viscous force remains almost the same.

Finally, Fig. 12 shows the mean thrust coefficient of the present model at different frequencies in comparison with the experimental and numerical results. In the low Mach number case, the thrust coefficient shows monotonic behaviour with frequency. However, the case with $Ma = 0.2$ shows a significantly different behaviour due to the effect of compressibility. We therefore conclude that pure pitching motion is even less efficient in producing thrust when the flow speed increases and the compressibility effects become important.
D. Transonic flow over NACA0012 airfoil in pitching motion

Finally, we solve a more challenging problem of a transonic flow over NACA0012 airfoil in pitching motion. Accurate computations of unsteady transonic flow is relevant in many applications such as wing flutter analysis or rotor-blade design.

We set the free-stream Mach number to $Ma = u_\infty / \sqrt{T_\infty} = 0.85$, with $T_\infty = 0.3$, Reynolds number $Re = 10000$, pitching amplitude $A = 2^\circ$ and reduced frequency of $k = 3$. Due to the high Mach number in this simulation, we need to employ the shifted lattices as presented in [27]. In our application, we use the lattice with a shift only in the free-stream direction as presented in [27].

In this way, deviations in the pertinent higher-order moments are minimized whenever the flow velocity is around $U_\infty$, and this transformation makes possible to simulate high Mach number flows [28]. For further details on the present model with shifted lattices the reader is referred to Saadat, Bösch, and Karlin [27].

$U = (U_x, U_y) = (0.3, 0)$. In this way, deviations in the pertinent higher-order moments are minimized whenever the flow velocity is around $U_\infty$, and this transformation makes possible to simulate high Mach number flows [28]. For further details on the present model with shifted lattices the reader is referred to Saadat, Bösch, and Karlin [27].
A motion of NACA0012 airfoil with FIG. 14. Time evolution of lift and drag coefficients for pitching downstream of the airfoil separation, and will also be influenced by the vortex shed-shock waves interact with the boundary layer, causing the flow of lambda-shocks appear further downstream as well. These reaches the boundary layer. Weak shock waves in the form accelerates causing a formation of weak oblique shock when it shear layers. Downstream of the leading edge, the flow interacts with the boundary and is observed that in this case, a complex flow field is formed with multiple shock waves interacting with the boundary and shear layers. The vortex shedding associated with the shear layer instabilities combines with the vortex shedding due to the airfoil movement, resulting in a complex vortex pattern in the wake region.

Time histories of lift and drag coefficients are presented in Fig. [13] Similar to the previous pitching cases with smaller Mach number, the average lift force is close to zero. However, there is a mean drag force of \( c_d \approx 0.0986 \) acting on the airfoil, which is significantly larger than its counterpart in low Mach number case. This confirms the previous observation about the effect of compressibility on increasing the drag force.

VI. CONCLUDING REMARKS

In this work, we proposed a solution methodology for the simulation of compressible flows on unstructured moving meshes based on the arbitrary Lagrangian-Eulerian (ALE) technique applied to a compressible lattice Boltzmann model. The kinetic equations of the compressible LB model on standard lattices were first mapped from a physical moving domain to a fixed computational domain. The resulting equations were solved by employing the second-order accurate finite element interpolation. It was shown, both theoretically and numerically, that the problem regarding the geometric conservation law (GCL), which needs special treatments in the ALE-Navier-Stokes solvers does not appear here and the proposed ALE-LB model satisfies the GCL condition exactly. The analysis of the model was conducted through simulation of compressible flow over NACA00012 airfoil undergoing plunging and pitching motions at different Mach numbers.

Most of the LB models in the literature employ a fixed background Cartesian grid, however, the present model is based on the body-fitted unstructured mesh which is more efficient in resolving small scale flow structures near the wall. Moreover, unlike previous LB studies, which were limited to low-speed incompressible flow, the LB model considered here is a compressible model which covers the range from subsonic to moderately supersonic regimes.

It was demonstrated that the model is able to properly predict the relevant features of the complex flow over flapping airfoil. In particular, the vortical patterns of wake, the time histories of lift and drag coefficients and their mean values agreed well with the experimental and numerical results in the literature. Both slow and fast plunging motion of airfoil produce a net mean thrust with very small average lift. Pitching motion, however, is not as effective as plunge motion and a thrust is generated at higher frequencies, and only when the compressibility effects are small. It was also observed that the impact of compressibility is mainly on the distribution of pressure force rather than the viscous force. Finally, in order to show the model’s performance in simulating high-speed flows, transonic flow over pitching airfoil was considered, where complex flow pattern involving multiple shock waves interacting with the boundary and shear layers were observed in the flow field.

The promising results of the proposed model open interesting prospects toward the numerical simulation of more complex flows such as the dynamic stall problem in compressible flows, flows involving multiple moving/deforming objects or fluid-solid interaction (FSI) problems. For problems including deformation or relative motion of multiple objects, a blending function is needed to construct the mapping function, as it was proposed in Persson, Bonet, and Peraire.[13] This would be the focus of our future research. Extension of the methodology to three dimensions is also another subject of our future works.

ACKNOWLEDGMENTS

This work was supported by the ETH research grant ETH-13 17-1 and the European Research Council (ERC) Advanced Grant No. 834763-PonD. The computational resources at the Swiss National Super Computing Center CSCS were provided under the grant s897.

The authors would like to thank Fabian Bösch and Benedikt Dorschner for useful discussions.

1 C. Bose and S. Sarkar, “Investigating chaotic wake dynamics past a flapping airfoil and the role of vortex interactions behind the chaotic transition,” Physics of fluids 30, 047101 (2018).
M. Boudreau, G. Dumas, M. Rahimpour, and P. Oshkai, “Experimental approach for the simulation of immersed moving solids with lattice boltzmann method,” Physical Review E 78, 446–472 (2013).

M. M. Koochesfahani, “Vortical patterns in the wake of an oscillating airfoil,” Physics of Fluids 620, 2773 (2013).

M. Berndt, J. Breil, S. Galera, M. Kucharik, P.-H. Maire, and M. Shashkov, “An arbitrary lagrangian–eulerian discontinuous galerkin method for simulations of flows over variable geometries,” Journal of Fluids and Structures 26, 312–329 (2010).

M. M. Koochesfahani, “Vortical patterns in the wake of an oscillating airfoil,” Physics of Fluids 620, 2773 (2013).

M. Boudreau, G. Dumas, M. Rahimpour, and P. Oshkai, “Experimental approach for the simulation of immersed moving solids with lattice boltzmann method,” Physical Review E 78, 446–472 (2013).

M. Berndt, J. Breil, S. Galera, M. Kucharik, P.-H. Maire, and M. Shashkov, “An arbitrary lagrangian–eulerian discontinuous galerkin method for simulations of flows over variable geometries,” Journal of Fluids and Structures 26, 312–329 (2010).

M. M. Koochesfahani, “Vortical patterns in the wake of an oscillating airfoil,” Physics of Fluids 620, 2773 (2013).

M. Berndt, J. Breil, S. Galera, M. Kucharik, P.-H. Maire, and M. Shashkov, “An arbitrary lagrangian–eulerian discontinuous galerkin method for simulations of flows over variable geometries,” Journal of Fluids and Structures 26, 312–329 (2010).

M. M. Koochesfahani, “Vortical patterns in the wake of an oscillating airfoil,” Physics of Fluids 620, 2773 (2013).

M. Berndt, J. Breil, S. Galera, M. Kucharik, P.-H. Maire, and M. Shashkov, “An arbitrary lagrangian–eulerian discontinuous galerkin method for simulations of flows over variable geometries,” Journal of Fluids and Structures 26, 312–329 (2010).

M. M. Koochesfahani, “Vortical patterns in the wake of an oscillating airfoil,” Physics of Fluids 620, 2773 (2013).

M. Berndt, J. Breil, S. Galera, M. Kucharik, P.-H. Maire, and M. Shashkov, “An arbitrary lagrangian–eulerian discontinuous galerkin method for simulations of flows over variable geometries,” Journal of Fluids and Structures 26, 312–329 (2010).

M. M. Koochesfahani, “Vortical patterns in the wake of an oscillating airfoil,” Physics of Fluids 620, 2773 (2013).
flows,” Computer Methods in Applied Mechanics and Engineering 157, 151–175 (1998).