Direct numerical simulation of turbulent pipe flow using the lattice Boltzmann method

Cheng Peng\textsuperscript{a,\*}, Nicholas Geneva\textsuperscript{a}, Zhaoli Guo\textsuperscript{b}, Lian-Ping Wang\textsuperscript{a,b}

\textsuperscript{a}Department of Mechanical Engineering, 126 Spencer Laboratory, University of Delaware, Newark, Delaware 19716-3140, USA
\textsuperscript{b}State Key Laboratory of Coal Combustion, Huazhong University of Science and Technology, Wuhan, P.R. China

Abstract

In this paper, we present a first direct numerical simulation (DNS) of turbulent pipe flow using the mesoscopic lattice Boltzmann method (LBM) on a D3Q19 lattice grid. DNS of turbulent pipe flow using LBM has never been reported previously, perhaps due to inaccuracy and numerical stability associated with the previous implementations of LBM in the presence of a curved solid surface. In fact, it was even speculated that the popular D3Q19 lattice might be inappropriate as a DNS tool for turbulent pipe flow. In this paper, we show, through a novel implementation, that accurate DNS simulation of turbulent pipe flow using the D3Q19 lattice is achievable. Our implementation makes use of an extended MRT model and a moving reference frame to enhance numerical stability. An issue associated with Galilean invariance in the moving frame is identified and resolved through coordinate transformation. The resulting turbulent flow statistics at a friction Reynolds number of \( Re_\tau = 180 \) are compared systematically with both published experimental and other DNS results based on solving the Navier-Stokes equations. The comparisons cover the mean-flow profile, the r.m.s. velocity and vorticity profiles, the mean and r.m.s. pressure profiles, the velocity skewness and flatness, and spatial correlations and energy spectra of velocity and vorticity. Overall, we conclude that our LBM method yields accurate results.

\*Corresponding author
1. Introduction

Over the last two decades, the lattice Boltzmann method (LBM) has been rapidly developed into an alternative and viable computational fluid dynamics (CFD) method for simulating viscous fluid flows involving complex boundary geometries or moving fluid-solid / fluid-fluid interfaces [1]. Unlike conventional (macroscopic) CFD methods that are based on directly solving the Navier-Stokes (N-S) equations, LBM is a mesoscopic method that is governed by a discretized version of gas kinetic equation in which molecular distribution functions are relaxed locally and then propagated to their neighboring locations. Since only local data communication is needed in each time step, LBM is extremely suitable for large-scale simulations that require parallel computation using a large number of processors, e.g., direct numerical simulations (DNS) of turbulent flows. Although still confined to relative low to moderate flow Reynolds (Re) numbers, DNS has been established as an independent research tool to study the physics of turbulent flows. The data generated from DNS not only agrees well with experimental results, but often provides greater details and insights of the flow field to a degree that may be very difficult or impossible to achieve experimentally.

In recent years, the capabilities of LBM as a DNS tool for turbulent flows have been explored by a series of studies in homogeneous isotropic turbulence and turbulent channel flows [2,3,4,5,6,7,8]. However, DNS of turbulent flow in a circular pipe using LBM has not yet been reported, to our best knowledge. So far all successful DNS studies of turbulent pipe flow have been performed using conventional numerical methods based on directly solving the N-S equations. Eggels et al. [9] presented the first DNS of a fully developed turbulent pipe flow using a second-order finite-volume (FV) method based on a uniform grid
in a cylindrical coordinates. The bulk Reynolds number based on the domain-averaged mean flow speed $U_b$ and the pipe diameter $D$ of their DNS is $Re_b = U_bD/\nu = 5300$, which corresponds to a Reynolds number based on the friction velocity $u_*$ and pipe radius $R$ of $Re_\tau = u_*R/\nu = 180$. Here $\nu$ is the fluid kinematic viscosity. The turbulent statistics from their simulation were reported to be in reasonable agreements with those from Laser Doppler Anemometry (LDA) and Particle Image Velocimetry (PIV) measurements at $Re_b = 5450$. They confirmed that the logarithmic layer deviates from the classical log-law at this low flow Reynolds number due to the pipe curvature effect. Loulou et al. [10] performed DNS of a fully developed turbulent pipe flow at $Re_b = 5600$ using a B-spline pseudo-spectral method. Compared with the previous FV simulations, Loulou et al.’s pseudo-spectral method is much more accurate providing likely a true benchmark for turbulent statistics, although later in this paper we will find that even the spectral results are not always accurate due to inadequate resolution and limited time duration of the simulation. Wagner et al. [11] investigated the effect of Reynolds number on the turbulent statistics of fully developed turbulent pipe flow by considering a range of flow Reynolds number from $Re_b = 5,300$ to 10,300, using a second-order FV method based on a non-uniform grid in the radial direction. The comparisons of turbulent statistics under different $Re_b$ suggest the strong influence of flow Reynolds number on both mean and fluctuation flow properties. Wu and Moin [12] conducted a DNS study of turbulent pipe flows with $Re_b$ up to 44,000 with a second-order FV method. More recently, Chin et al. [13] conducted DNS of turbulent pipe flow at $Re_\tau = 170$ and 500 with different pipe length using spectral method. Their findings suggest the pipe lengths for converged turbulent statistics are different in the inner and outer regions in a turbulent pipe flow. For inner region (near wall region), the sufficient pipe length should scale with the wall unit and at least on the order of $O(6300)$, while for the outer region, the turbulent statistics convergence with a pipe length that is approximately $4\pi D$. El Khoury et al. [14], performed a series of DNS in a turbulent pipe flow with different Reynolds numbers, $Re_\tau = 180, 360, 550$ and 1000. Based on their results, a comparison
study was also carried out among turbulent flow statistics in a channel, pipe and boundary layer. Their work indicates that the pressure exhibits the most significant difference among the three wall-bounded turbulent flows.

Since turbulent pipe flow represents a more realistic flow often encountered in engineering applications, it is highly desirable to perform DNS of this flow with LBM in order to establish LBM as a viable alternative for turbulent flow simulation. For LBM simulations, the presence of a curved pipe wall adds a great deal of fundamental difficulties. The accurate representation of the curved pipe surface is essential since the majority of turbulent kinetic energy production and viscous dissipation is confined to the near-wall region. Unlike conventional CFD methods, such as the finite volume (FV) scheme, that can be applied to a computational mesh in cylindrical coordinates which allows accurate representation of the pipe wall [9, 10, 11, 12], standard LBM is restricted to a cubic grid in Cartesian coordinates that does not conform with the cylindrical surface. Thus to resolve the pipe boundary in LBM, local interpolation is used which could introduce both inaccuracy and numerical instability. Although the boundary treatments on a curved boundary have received a great deal of attention in LBM [15, 16, 17], their impacts on the accuracy and numerical instability in a turbulent pipe flow simulation has not been systematically studied.

Previously, only large-eddy simulations (LES) of turbulent pipe flow have been attempted using LBM [6, 18], which confirmed the existence of so-called lattice effects in the simulation of turbulent flow in circular pipe due to the lack of full isotropy of a D3Q19 lattice [19]. Kang and Hassan [6] conducted LES of turbulent pipe flow on both D3Q19 and D3Q27 lattice grids. Their results indicate that with the same resolution, the mean and fluctuation velocities with D3Q27 are in reasonable agreement with the previous DNS benchmark [9], while those with D3Q19 developed an unphysical secondary flow pattern and exhibited large deviations (up to 28% relative error) from the respective benchmark [6]. Suga et al. [18] later presented a similar LES of turbulent pipe flow based on their proposed multi-relaxation time (MRT) LBM model with D3Q27 lattice grid. Both their mean flow velocity and root-mean-squared (r.m.s.) velocity profiles are in
reasonable agreement with the corresponding DNS benchmark [9]. As a result of
the issues associated the D3Q19 lattice, these studies have argued that D3Q27
has better isotropy properties [6, 18] and thus should be adopted instead of
D3Q19 in DNS of a turbulent pipe flow. However, careful benchmarking of the
LES simulation results is still necessary in order to fairly assess the capabilities
of different lattice models in LBM. As will become clear later, the capabilities
and accuracy of a lattice model depend largely on the implementation details
used to meet the no-slip boundary condition at the fluid-solid interface. In this
paper, we will introduce a novel implementation of LBM, to demonstrate that
accurate DNS of a turbulent pipe flow is actually achievable even on a D3Q19
lattice.

We should note that there are some limited efforts to introduce cylindrical
coordinates into LBM. One typical approach is the development of LBM mod-
els for axisymmetric flows [20, 21, 22]. The idea is to add a source term in
the lattice Boltzmann equation (LBE) to reproduce the N-S equations in the
cylindrical coordinates, when the flow has no azimuthal dependence. Although
such methods allows one to simulate three dimensional axisymmetric flow on
a two-dimensional lattice grid, which greatly enhances the computational effi-
ciency, it cannot be used for three-dimensional turbulent pipe flow simulation.
Other approaches that could be relevant to DNS of turbulent pipe flow is the
development of finite-difference (FD) [23, 24], FV [25, 26, 27, 28] or spectral
implementations [29] of the discrete-velocity Boltzmann equation, by uncou-
plying the discretization of physical space and lattice time. Those alternative
Boltzmann-equation based approaches do allow the use of non-uniform grid in
curvilinear coordinates to fit, for example, the circular pipe wall. However,
compared with the standard LBM that uses direct streaming, these methods
introduce additional diffusion and dissipation in the discretization of space and
time, which may affect the accuracy of the simulated turbulent statistics. More-
ever, they are less efficient in terms of parallel scalability when compared to the
standard LBM.

The present work has three major objectives. First, we will develop a novel
LBM implementation on a D3Q19 lattice and use it as a DNS tool for turbulent pipe flow. In order to achieve this goal, we have introduced improved methods both within the LBM scheme itself and at the fluid-solid interface to address issues related to accuracy and numerical instability in the presence of a curved wall. While the study here is limited to turbulent pipe flow, the methods we will incorporate have broader implications for simulating turbulent flows in complex geometries.

Second, we will conduct a careful comparison of the resulting turbulent statistics between our LBM and other conventional methods [9, 11, 10] as well as those from experimental measurements. Experimental data of turbulent pipe flow at moderate flow $Re$ numbers include measurements using LDA [9, 30] and PIV [31]. Such systematic comparisons will not only provide a fair assessment on the performance of our LBM, especially its boundary treatment schemes on the simulation of flow in a cylindrical geometry; but will also reveal inaccuracies in some previous DNS and experimental data which should benefit the community in the future.

The rest of the paper is arranged as follows. In Sec. 2, we discuss the physical problem and the numerical method with all novel implementation details. Those details include a MRT model with extended equilibria based on a D3Q19 lattice grid to enhance the numerical stability, a non-uniform perturbation force field to excite the turbulence and a new bounce back scheme to remove the Galilean-invariance errors when the flow system is solved in a purposely-designed moving reference frame. In Sec. 3, we first validate the method against the transient laminar pipe flow and briefly discuss the accuracy of the LBM method. Then, in Sec. 4 the turbulent flow statistics will be discussed systematically through a careful comparison with existing numerical and experimental results. A large number of flow statistics in both physical space and spectral space will be analyzed. Finally, Sec. 5 contains a summary and the main conclusions of the present work. The Appendix explains how the local vorticity is computed, and the necessary additional considerations near the curved pipe wall to convert data from a Cartesian grid to flow field in cylindrical coordinates.
2. Problem statement and the simulation method

We consider the classical flow inside a circular pipe driven by a constant external force. As sketched in Fig. 1(a), $r$, $\theta$ and $z$ represent the radial, azimuthal and streamwise directions, respectively. The radius of the pipe is $R$, the length of the pipe is $L$. Periodic boundary condition is assumed in the streamwise direction, while on the pipe wall, the no-slip boundary condition is applied.

A constant body force $g$ (per unit mass), or equivalently a mean pressure gradient is applied in the streamwise direction to drive the flow. At the fully developed stage and averaged over time, force balance between the driving and the viscous force is established as $2\pi RL\langle \tau_w \rangle = \pi R^2 L \rho g$, which leads to the expression of the averaged wall shear stress $\langle \tau_w \rangle$ and the friction velocity $u^*$ as

$$\langle \tau_w \rangle = \frac{1}{2} \rho g R, \quad u^* = \sqrt{\frac{\langle \tau_w \rangle \rho}{\rho}} = \sqrt{\frac{gR}{2}}. \quad (1)$$

where $\rho$ is the fluid density. The frictional Reynolds number can then be defined as $Re_\tau = u^* R / \nu = R / (\nu / u^*)$, where $\nu$ is the kinematic viscosity, $y^* = \nu / u^*$ is characteristic length scale of the viscous sublayer, which is known as the wall unit. The large-scale eddy-turnover time is defined as $R / u^*$.

In this study, we set $Re_\tau = 180$, in order to benchmark our results with the database from the other published studies based on the pseudo-spectral (PS) and finite-volume (FV) methods. The bulk $Re$ number, defined with the mean flow velocity magnitude $\bar{U}$ and pipe diameter $D$, is around 5300. The velocity
scale ratio is $\bar{U}/u^* \approx 14.7$.

2.1. The lattice Boltzmann method

Unlike conventional or macroscopic CFD methods that solve the incompressible Navier-Stokes equations, LBM amounts to solving the weakly compressible N-S equations with a model speed of sound $c_s$ ($c_s = 1/\sqrt{3}$ in the lattice units). Such feature confines the maximum flow speed in order to ensure that the maximum local Mach number is small. Under this constraint, to achieve high $Re$ with a manageable mesh size, the viscosity $\nu$ in the simulation needs to be very small. Since the fluid viscosity is related to the relaxation parameter in the LBM, small viscosities may lead to severe numerical instability.

Based on this consideration, an extended MRT LBM on a D3Q19 lattice is adopted in the present study. Compared with the single-relaxation time (SRT) collision operator, the MRT collision model is preferred due to its better numerical stability, as the relaxation parameters associated with moments irrelevant to the N-S equations can be optimized for this purpose [32, 33]. The evolution equation of the MRT LBM reads as

$$f(x + e_i \delta_t, t + \delta_t) - f(x, t) = -M^{-1}S \left[ m(x, t) - m^{eq}(x, t) \right] + M^{-1}\Psi,$$  \hspace{1cm} (2)

where $f$ is the distribution function vector, $t$ and $x$ are the time and spatial coordinate, respectively. $e_i$ is the lattice particle velocity in the $i$ direction. In D3Q19, nineteen three-dimensional lattice velocities are used and they are

$$e_i = \begin{cases} 
(0, 0, 0) c, & i = 0, \\
(\pm 1, 0, 0) c, (0, \pm 1, 0) c, (0, 0, \pm 1) c, & i = 1, 2, \ldots, 6, \\
(\pm 1, \pm 1, 0) c, (\pm 1, 0, \pm 1) c, (0, \pm 1, \pm 1) c, & i = 7, 8, \ldots, 18.
\end{cases}$$  \hspace{1cm} (3)

where $c = \delta_x/\delta_t$, $\delta_x$ and $\delta_t$ are the grid spacing and time step size, respectively. Nineteen independent moments are defined by $m = Mf$, where $M$ is a $19 \times 19$ transform matrix. The diagonal matrix $S$ contains relaxation parameters and is written as

$$S = \text{diag}(0, s_e, s_x, s_j, s_q, s_j, s_q, s_j, s_q, s_{\nu}, s_{\nu}, s_{\pi}, s_{\pi}, s_{\nu}, s_{\nu}, s_{\nu}, s_{m}, s_{m}, s_{m}).$$  \hspace{1cm} (4)
In the standard three-dimensional MRT LBM, the values of the relaxation parameters for the energy and stress modes, i.e., \( s_e \) and \( s_\nu \), are directly related to the bulk and shear viscosities \( \nu^V \) and \( \nu \) of the continuum fluid, as

\[
\nu^V = \frac{2}{9} \left( \frac{1}{s_e} - \frac{1}{2} \right) c^2 \delta t, \quad \nu = \frac{1}{3} \left( \frac{1}{s_\nu} - \frac{1}{2} \right) c^2 \delta t.
\]  

The values of those relaxation parameters are between 0 and 2 in order to maintain positive viscosities. Numerical instability may arise when those relaxation parameters approach the two bounding values.

The vector \( \mathbf{m}^{(eq)} \) in Eq. (2) specifies the equilibrium moments. Normally, in the standard MRT LBM, \( \mathbf{m}^{(eq)} \) are functions of only the conserved moments, i.e., density fluctuation \( \delta \rho \) and momentum \( \rho_0 \mathbf{u} \) as in [33] and [4]. However, as indicated by Eq. (5), small viscosities in the high Re flows push the relaxation parameters \( s_e \) and \( s_\nu \) to their upper limit, which sabotages the numerical stability of the simulation. In order to address this problem, stress components are introduced into \( \mathbf{m}^{(eq)} \) to modify Eq. (5). This idea was firstly proposed by [35] with a SRT collision model. Following the same idea, the equilibrium moments in our extended MRT model is formally written as

\[
\mathbf{m}^{(eq)} = \mathbf{m}^{(eq,0)} + \mathbf{m}^{(eq,1)},
\]  

with

\[
\begin{align*}
\mathbf{m}^{(eq,0)} &= \delta \rho, \quad \mathbf{m}^{(eq,0)}_1 = -11 \delta \rho + 19 \rho_0 (u^2 + v^2 + w^2), \\
\mathbf{m}^{(eq,0)}_2 &= \alpha \delta \rho + \beta \rho_0 (u^2 + v^2 + w^2), \quad \mathbf{m}^{(eq,0)}_3 = \rho_0 u, \quad \mathbf{m}^{(eq,0)}_4 = -2 \rho_0 u / 3, \\
\mathbf{m}^{(eq,0)}_5 &= \rho_0 v, \quad \mathbf{m}^{(eq,0)}_6 = -2 \rho_0 v / 3, \quad \mathbf{m}^{(eq,0)}_7 = \rho_0 w, \quad \mathbf{m}^{(eq,0)}_8 = -2 \rho_0 w / 3, \\
\mathbf{m}^{(eq,0)}_9 &= \rho_0 (2u^2 - v^2 - w^2), \quad \mathbf{m}^{(eq,0)}_{10} = \gamma \rho_0 (2u^2 - v^2 - w^2), \\
\mathbf{m}^{(eq,0)}_{11} &= \rho_0 (v^2 - w^2), \quad \mathbf{m}^{(eq,0)}_{12} = \gamma \rho_0 (2u^2 - v^2 - w^2), \quad \mathbf{m}^{(eq,0)}_{13} = \rho_0 uv, \\
\mathbf{m}^{(eq,0)}_{14} &= \rho_0 uw, \quad \mathbf{m}^{(eq,0)}_{15} = \rho_0 vw, \quad \mathbf{m}^{(eq,0)}_{16} = \mathbf{m}^{(eq,0)}_{17} = \mathbf{m}^{(eq,0)}_{18} = 0.
\end{align*}
\]  

where \( \alpha, \beta \) and \( \gamma \) are free parameters that are irrelevant to the N-S equations.

\(^1\)Note that, here we follow the incompressible LBM model proposed by [34], partitioning the density into a local density fluctuation \( \delta \rho \) and a constant background density \( \rho_0. \)
The sequence of the moments and their definitions are identical with those in [33]. Furthermore, we choose \( \alpha = 0, \beta = -\frac{475}{63} \) and \( \gamma = 0 \) as in [33].

The second term \( \mathbf{m}^{(eq,1)} \) in Eq. (6) contains additional equilibrium moments being introduced to modify Eq. (5). Only 6 of the 19 elements in \( \mathbf{m}^{(eq,1)} \) are relevant to the derivation of the N-S equations, and they are

\[
\begin{align*}
\mathbf{m}_1^{(eq,1)} &= \rho_0 \zeta (\partial_x u + \partial_y v + \partial_z w), \\
\mathbf{m}_9^{(eq,1)} &= \rho_0 \lambda (4\partial_x u - 2\partial_y v - 2\partial_z w), \\
\mathbf{m}_{11}^{(eq,1)} &= \rho_0 \lambda (2\partial_y v - 2\partial_z w), \\
\mathbf{m}_{13}^{(eq,1)} &= \rho_0 \lambda (\partial_x v + \partial_y u), \\
\mathbf{m}_{14}^{(eq,1)} &= \rho_0 \lambda (\partial_y w + \partial_z v), \\
\mathbf{m}_{15}^{(eq,1)} &= \rho_0 \lambda (\partial_z u + \partial_x w)
\end{align*}
\]

and the others are simply set to zero. Using the Chapman-Enskog expansion, we can show that this extended model leads to the Navier-Stokes equations with the following shear and bulk viscosities [36]

\[
\nu = \frac{1}{3} \left( \frac{1}{s_\nu} - \frac{1}{2} \right) c^2 \delta_t - \lambda, \quad (9a)
\]

\[
\nu^V = \frac{2}{9} \left( \frac{1}{s_e} - \frac{1}{2} \right) c^2 \delta_t - \zeta. \quad (9b)
\]

Therefore, the introduction of \( \mathbf{m}^{(eq,1)} \) into the equilibrium moments modifies the relationships between viscosities and corresponding relaxation parameters from Eq. (5) to Eq. (9). Even for small physical viscosities, there is no need to set the relaxation parameters \( s_\nu \) and \( s_e \) too close to 2, due to the presence of non-zero \( \lambda \) and \( \zeta \). In the actual implementation, we find that the best stability can be achieved with \( 1.8 \leq s_\nu \leq 1.9 \), while the value of \( s_e \) is found to have no obvious effect on the numerical stability.

The mesoscopic external force vector \( \mathbf{\Psi} \) in Eq. (2) can be designed via an inverse design analysis in the moment space, as done in [37]. The relevant
elements of $\Psi$ in terms of the N-S equations are summarized as

$$\Psi_1 = 38 \left(1 - 0.5s_e\right) (uF_x + vF_y + wF_z),$$
$$\Psi_3 = (1 - 0.5s_j) F_x,$$
$$\Psi_5 = (1 - 0.5s_j) F_y,$$
$$\Psi_7 = (1 - 0.5s_j) F_z,$$
$$\Psi_9 = 2 \left(1 - 0.5s_u\right) \left(vF_x + uF_y\right),$$
$$\Psi_{11} = 2 \left(1 - 0.5s_u\right) \left(vF_z + wF_y\right),$$
$$\Psi_{13} = \left(1 - 0.5s\right) \left(uF_z + wF_x\right),$$
$$\Psi_{15} = \left(1 - 0.5s\right) (uF_z + wF_x).$$

(10)

where $u = (u, v, w)$ is the local fluid velocity, and $F = (F_x, F_y, F_z)$ is the local macroscopic force per unit volume. The other mesoscopic terms in $\Psi$ are irrelevant and can be set to zero for simplicity.

It should be noted that all the relevant $\mathbf{m}^{(eq,1)}$ components in Eq. (8) can be calculated mesoscopically as

$$\rho_0 (\partial_x u + \partial_y v + \partial_z w) = \frac{G_1}{\left(\zeta - \frac{38c^2\delta_h}{3s_e}\right)}$$

(11a)

$$\rho_0 (4\partial_x u - 2\partial_y v - 2\partial_z w) = \frac{G_9}{\left(\lambda - \frac{c^2\delta_h}{3s\nu}\right)}$$

(11b)

$$\rho_0 (2\partial_y v - 2\partial_z w) = \frac{G_{11}}{\left(\lambda - \frac{c^2\delta_h}{3s\nu}\right)}$$

(11c)

$$\rho_0 (\partial_x v + \partial_y u) = \frac{G_{13}}{\left(\lambda - \frac{c^2\delta_h}{3s\nu}\right)}$$

(11d)

$$\rho_0 (\partial_y w + \partial_z v) = \frac{G_{14}}{\left(\lambda - \frac{c^2\delta_h}{3s\nu}\right)}$$

(11e)

$$\rho_0 (\partial_z u + \partial_x w) = \frac{G_{15}}{\left(\lambda - \frac{c^2\delta_h}{3s\nu}\right)}$$

(11f)

where $G_i = M_{ij}f_j - m^{(eq,0)}_i + \Psi_i/(2 - s_i)$. The above relations ensure that all quantities can be computed mesoscopically to maintain the second-order accuracy of the extended LBM.

2.2. Parameter set-up

To resolve the smallest scale in the flow, it usually requires that inside the viscous sublayer $((R - r)/y^* \leq 8)$, there are at least three grid points in the
radial direction \[3\], \textit{i.e.}, \(\delta r^+ < 2.5\) for uniform grid (hereafter we use the superscript + to denote quantities normalized by the wall unit \(y^*\) or friction velocity \(u^*\)). With this constraint, we choose \(R = 148.5\) in lattice units, which implies that \(\delta r^+ = 180/148.5 \approx 1.212\). This grid spacing is sufficiently small to resolve all scales even for the near wall region. On the other hand, the pipe length in the streamwise direction must be long enough to minimize the effect the periodic boundary condition \[13\]. In the present work, we set \(L = 1799\), which is \(12.11R\). This pipe may not be long enough to obtain converged turbulent statistics near the pipe wall, but it allows fair comparisons between our present simulation and those existing datasets, which were obtained with a pipe length of about \(10R\) \[9\] \[11\] \[10\]. While the physical problem is stated in a cylindrical coordinate, \textit{i.e.}, \(r, \theta, z\) coordinates, the LBM simulation is set up in the Cartesian coordinates, \textit{i.e.}, \(x, y\) and \(z\). Based on the two aforementioned aspects, the grid resolution is chosen to be \(N_x \times N_y \times N_z = 300 \times 300 \times 1799\). The whole computation domain is decomposed in the \(x\) and \(z\) directions as \(90 \times 15\) subdomains using 2D domain decomposition as shown in Fig. 1(b), as we did in our previous studies \[7\] \[8\]. The data communication between neighboring domains is handled with Message Passing Interface (MPI).

The pipe center is located in the \(x - y\) plane at \(r_c = (150.5, 150.5)\), which is slightly off the domain center to suppress the secondary flow patterns observed in \[19\], \[6\] and \[18\]. The shifting of the pipe center relative to lattice nodes breaks the symmetry of the boundary link configuration, thus reducing the spurious secondary flow due to the D3Q19 lattice symmetry. Though it has been suggested that the D3Q27 lattice has a better isotropy leading to a much weaker unphysical secondary flow in a circular pipe, our LBM simulation is still based on the D3Q19 lattice because its capability in simulating turbulent flows has been previously confirmed \[see 2\] \[3\] \[5\] \[7\] \[8\]. One novel aspect of this paper is to precisely demonstrate that the D3Q19 lattice is adequate for DNS of turbulent pipe flow. The kinematic viscosity \(\nu\) is chosen to be 0.0032, which is above the limiting value 0.00254 \[33\]. The bulk viscosity \(\nu^V\) is set as 1.0 (both viscosities are in lattice unit) in order to dissipate the acoustic waves in the
Table 1: Physical parameters used for the simulation of turbulent pipe flow

<table>
<thead>
<tr>
<th>( N_x \times N_y \times N_z )</th>
<th>( \nu )</th>
<th>( \nu^{V} )</th>
<th>( R )</th>
<th>( u^{*} )</th>
<th>( Re_{p} )</th>
<th>( \delta x/y^{*} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>300 \times 300 \times 1799</td>
<td>0.0032</td>
<td>1.0</td>
<td>148.5</td>
<td>0.00388</td>
<td>180</td>
<td>1.212</td>
</tr>
</tbody>
</table>

Table 2: LBM model parameters used in the simulations

<table>
<thead>
<tr>
<th>( \alpha )</th>
<th>( \beta )</th>
<th>( \gamma )</th>
<th>( s_c )</th>
<th>( s_c )</th>
<th>( s_j )</th>
<th>( s_{q} )</th>
<th>( s_{\nu} )</th>
<th>( s_{m} )</th>
<th>( \zeta )</th>
<th>( \lambda )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>-475/63</td>
<td>0</td>
<td>1.0</td>
<td>1.5</td>
<td>1.0</td>
<td>1.8</td>
<td>1.8</td>
<td>1.5</td>
<td>1.5</td>
<td>-0.889</td>
</tr>
</tbody>
</table>

flow [38]. The use of a large bulk viscosity may contaminate the pressure field, which must be corrected, as explained in Section 4.3. The key parameters used in the present study are summarized in Table 1 and Table 2.

2.3. Flow initialization and prescribed excitation at the early stage

The flow is initialized with the following mean velocity profile containing the viscous sublayer profile and inertial sublayer (or logarithmic) profile

\[
U^{+} (\delta^{+}) = \begin{cases} 
\delta^{+}, & \text{if } \delta^{+} \leq 10.8, \\
\frac{1}{6.4} \ln (\delta^{+}) + 5.0, & \text{if } \delta^{+} > 10.8.
\end{cases}
\]  

(12)

where \( \delta = R - r \) is the distance from the pipe wall. It should be noted that, although in a fully developed turbulent pipe flow, the mean velocity profile does not strictly conform with Eq. (12) in the logarithmic region [9, 30, 11], the developed flow mean velocity profile should not be affected by this initial condition. While this initial velocity field is not a solution of the N-S equations, it serves the purpose of speeding up the flow transition to realistic, fully developed turbulent pipe flow when compared to starting the flow from rest. The initial distribution functions are simply given as the leading-order equilibrium distributions, i.e., \( f = M^{-1} m^{(eq,0)} \), based on the initial velocity field and zero initial pressure.
The flow is forced by the body force \((F_r, F_\theta, F_z) = (0, 0, \rho_0 g)\). To accelerate the transition from the initial laminar flow to turbulence, we add a non-uniform, divergence-free force field to the flow only during the first three large-eddy turnover times of the simulation, namely,

\[
F'_r = -g\kappa B_0 \frac{R k_z l}{r} \sin \left( \frac{2\pi t}{T} \right) \left\{ 1 - \cos \left( \frac{2\pi (R - r - l_0)}{l} \right) \right\} \cos \left( k_z \frac{2\pi z}{L} \right) \cos (k_\theta \theta)
\]

\[
F'_\theta = g (1 - \kappa) B_0 \frac{k_z 2\pi R}{k_\theta} \sin \left( \frac{2\pi t}{T} \right) \sin \left( \frac{2\pi (R - r - l_0)}{l} \right) \cos \left( k_z \frac{2\pi z}{L} \right) \sin (k_\theta \theta)
\]

\[
F'_z = -g B_0 \frac{R}{r} \sin \left( \frac{2\pi t}{T} \right) \sin \left( \frac{2\pi (R - r - l_0)}{l} \right) \sin \left( k_z \frac{2\pi z}{L} \right) \cos (k_\theta \theta)
\]

(13a) (13b) (13c)

where \(k_z\) and \(k_\theta\) are the wavenumbers of the perturbation force in streamwise and azimuthal directions, respectively. \(T\) is the forcing period, \(B_0\) is the forcing magnitude, \(\kappa\) is the weighting parameter that distributes the perturbation in radial and azimuthal directions. The above forcing is added only to the region \(l_0 \leq R - r \leq l_0 + l\) in radial direction. After the initial period of three eddy turnover times, the perturbation forcing is no longer applied, namely, \((F'_r, F'_\theta, F'_z) = (0, 0, 0)\) for \(t \geq 3R/u^*\).

In this paper, since we focus only on the fully developed stage of the turbulent pipe flow, how to drive the flow to its stationary turbulence can be flexible. To save computational resources, the simulation was started in a smaller domain with \(N_{0x} \times N_{0y} \times N_{0z} = 300 \times 300 \times 599\), for about 30 eddy turnover times, to generate a preliminary turbulent flow field. For the the starting smaller domain, the parameters in Eq. (13) are chosen as: \(k_z = 3, k_\theta = 2, T = 2000, l_0 = 0.2R, l = 0.4R, B_0 = 50.0, \kappa = 0.5\). At this stage, the flow is already fully developed, but for a 1/3 pipe length. We then copied this flow twice to form a new starting flow field. Since the streamwise domain size 1799 is not an exact multiple of 599, the missing information for two gap layers (with grid indices \(n_z = 600\) and...
between copies is filled by a simple linear extrapolation scheme, as

\[
\begin{align*}
    f_i (n_x, n_y, 600) &= 2f_i (n_x, n_y, 599) - f_i (n_x, n_y, 598), \\
    f_i (n_x, n_y, 1200) &= 2f_i (n_x, n_y, 1199) - f_i (n_x, n_y, 1198).
\end{align*}
\]

Once the copy and extension are done, the LBM simulation was run at least another 10 eddy turnover times for the flow to reach the stationary stage in the full domain.

2.4. The use of a moving frame and improved implementation at the pipe wall

The interpolated bounce back rule is applied on the pipe wall to maintain the overall second-order accuracy in the simulation. While both linear and quadratic interpolated schemes possess at least the second-order spatial accuracy, they have significant difference in terms of numerical stability of the present simulation. Although the quadratic interpolation schemes have generally better accuracy than the corresponding linear interpolation schemes [see 39], they are not numerically stable in the present pipe-flow simulation. The reason could be related to the appearance of negative coefficient in front of the distribution functions on the furthest node point in the quadratic interpolation [15, 17]. Furthermore, consider the very thin boundary layer and the large velocity gradient in the near-wall region, linear interpolation schemes are more preferable than the quadratic schemes due to their better localization. For these reasons, in the present simulation, the linear interpolated bounce-back schemes by [15] and [17] are applied, and both of them are found to be numerically stable.

The checkerboard instability was another major problem encountered in the present simulation. Unlike the instability results from the quadratic interpolation that originates from the pipe wall, the checkerboard instability appears first in the pipe center region and quickly propagates to the whole flow field, as shown in Fig. 2. The checkerboard instability is mainly due to the insufficient discretization of the mesoscopic velocity field that causing the distribution functions at one node point to isolate from its surrounding neighbors [40]. Normally, such checkerboard instability can be eliminated by applying odd numbers
of grid points in all location, as indicated in [40] and in [8]. However, this simple solution does not apply to the current pipe flow simulation. We tested both 599 and 600 lattice points in the streamwise direction, but the same checkerboard pattern was observed. In our simulation, the checkerboard instability is found to be strongly related to the Mach number ($Ma$) in the center region of the pipe, which puts a much more critical confinement on the maximum allowed $Ma$ in the pipe flow simulation than in the our previous channel flow simulations with the same $Re_\tau$ [8].

To reduce the local $Ma$ number in the center region, a negative (opposite to the driving force or mean flow) reference velocity is added to the whole system, namely, we force the pipe wall to move at $u_w = -12u^*$. Physically, this constant translation velocity of the moving reference frame shall have no effect on the simulation results. Unfortunately, we discovered that the principle of Galilean invariance can be violated [see 41], as explained below. In the turbulent pipe flow simulation, a careful investigation indicated that the Violation of Galilean Invariance (VGI) errors originate from two aspects, the very non-uniform flow near the pipe wall and the use of interpolated bounce back. For the configuration shown in Fig. 3, the momentum change $\delta J$ of the fluid phase at the boundary node $x_b$ due to the interaction with the wall can be written as

$$\delta J = \sum_{\text{links}} (f_i e_i - f_i e_i)$$

(15)

where $f_i$ is the incident distribution that disappears on the solid surface and $f_i$ is the distribution function constructed via a boundary treatment scheme. $f_i$ and $f_i$ at the boundary node $x_b$ contains two parts, the equilibrium part that can be defined in terms of the conserved moments [4] namely, the density and velocity at the wall, and the non-equilibrium part that can be approximated as only a function of stress components [42, 43]. Since the latter has no dependence on the reference velocity in the system, we can tentatively put that aside from

---

2Note that, in our extended MRT model, the equilibrium part is consist of both conserved moments and stress components, but the similar arguments remain true.
Figure 2: The pressure pattern caused by the checkerboard instability. (a) pressure contour in a cross section with Bouzidi et al.’s linear interpolated bounce back, (b) same as (a) with Yu et al.’s linear interpolated bounce back, (c) pressure distribution on a line along the streamwise direction, (d) pressure distribution on a line along the radial direction. The grid resolution is $N_x \times N_y \times N_z = 300 \times 300 \times 599$. 
our discussion for now. Substituting the equilibrium part of \( f_i \) and \( f_{\bar{i}} \), which usually read as

\[
\begin{align*}
  f_i^{(eq)} &= w_i \delta \rho + \rho_0 w_i \left[ \frac{\left( e_i \cdot u_w \right)}{c_s^2} + \frac{\left( e_i \cdot u_w \right)^2}{2c_s^4} - \frac{\left( u_w \cdot u_w \right)}{2c_s^2} \right], \quad (16a) \\
  f_{\bar{i}}^{(eq)} &= w_{\bar{i}} \delta \rho + \rho_0 w_{\bar{i}} \left[ \frac{\left( e_{\bar{i}} \cdot u_w \right)}{c_s^2} + \frac{\left( e_{\bar{i}} \cdot u_w \right)^2}{2c_s^4} - \frac{\left( u_w \cdot u_w \right)}{2c_s^2} \right]. \quad (16b)
\end{align*}
\]

into Eq. (15), and considering \( e_i = -e_{\bar{i}}, w_i = w_{\bar{i}} \), we obtain

\[
\delta J = \sum_{\text{B links}} \left\{ -2w_i \delta \rho e_i - 2\rho_0 w_i \left[ \frac{\left( e_i \cdot u_w \right)^2}{2c_s^4} - \frac{\left( u_w \cdot u_w \right)}{2c_s^2} \right] e_i \right\} \quad (17)
\]

where the summation is over all boundary links at the boundary node. The dependence of the second term on the wall velocity \( u_w \) indicates that the momentum change contributed by a boundary link is not Galilean invariant.

Ideally, such VGI error emerges from link \( i \) is expected to cancel with the corresponding error from link \( j \) (see Fig. 3), which can be shown if the simple mid-link bounce-back is applied to all boundary links. However, when the interpolated bounce back scheme is applied, the precise cancellation is unlikely since bounced-back distributions \( f_i \) and \( f_{\bar{j}} \) depend not only on the information at \( x_b \), but also \( x_{f,i} \) and \( x_{f,j} \) (see Fig. 3), respectively. Therefore, the VGI error at \( x_b \) becomes non-trivial. As shown in [41], the VGI error creates an additional drag that affects the balance between the driving force and the true wall viscous...
force, as a result the simulated mean flow velocity becomes inaccurate (typically smaller). To solve this problem, we developed a new bounce back scheme using the idea of coordinate transformation. The essential idea is to always perform the bounce back operation in the frame moving with the wall. The procedure for this new bounce-back scheme with the moving pipe wall is as follows:

1. After the collision sub-step, starting with the distribution functions in the *fixed* coordinate system (*i.e.*, the coordinate system attached to the lattice grid), we construct the distribution functions in the coordinate system moving with the wall (the *moving* coordinate system), for all distribution functions that reach the boundary fluid node \( x_b \) after propagation. These distribution functions include two sets, the distribution functions that arrive \( x_b \) from direct propagation (including the one at rest), and those from bounce-back. For the first group, we have

\[
f'_i(t + \delta t, x_b) = f'^*_{i}(t, x_b - e_i)
\]  

where \( f \) with superscript prime indicates the distribution functions in the *moving* coordinate system while \( * \) indicates the post-collision distribution functions. For the second set of the distribution functions, certain bounce-back scheme is applied. For demonstrative purpose, we use the double linear interpolation scheme proposed by Yu et al. \[17\]

\[
f'_i(t + \delta t, x_b) = \frac{q}{1 + q} f'^*_{i}(t, x_b) + \frac{1 - q}{1 + q} f'^*_{i}(t, x_f) + \frac{q}{1 + q} f'^*_{i}(t, x_b)
\]

(19)

All the involving post-collision distribution functions in the *moving* coordinate system are transformed from the *fixed* coordinate system as

\[
f'^*_{i}(t, x) = f'^{(neq)*}_{i}(t, x; u^*, \delta \rho^*) + f'^{(eq)*}_{i}(t, x; u^* - u_w, \delta \rho^*)
\]

(20)

where \( u^* \) and \( \delta \rho^* \) are post-collision local velocity and density fluctuation at the location \( x \).

2. Next, use \( f'_i(t + \delta t, x_b) \) to update the density fluctuation and velocity at
\( x_b \) in the moving coordinate system

\[
\delta \rho' (t + \delta t, x_b) = \sum_i f_i' (t + \delta t, x_b),
\]

\[
\rho_0 u' (t + \delta t, x_b) = \sum_i f_i' (t + \delta t, x_b) e_i.
\]

(21)

3. Finally, transform all distribution functions at \( x_b \) in the moving coordinate system back to the fixed coordinate system, as

\[
f_i (t + \delta t, x_b) = f_i'_{eq} (t + \delta t, x_b; u' + u_w, \delta \rho') + f_i'_{neq} (t + \delta t, x_b; u', \delta \rho').
\]

(22)

Since in Step 3 all distribution functions are updated, the momentum exchange between solid and fluid phases at a boundary node contains two parts. The first part is realized via momentum exchange during the bounce back. At the same time, the momentum carried by the direct streaming from the neighboring nodes could also have been modified. This part of momentum change should be taken into consideration in order to obey the Newton’s Third Law locally. In summary, the hydrodynamic force acting on the solid surface should be calculated as

\[
F (x_b, t) \delta \tau = \sum_{B \text{ links}} \left[ f_i (t + \delta t, x_b) + f_i' (t, x_b) \right] e_i + \sum_{\text{others}} \left[ f_i (t + \delta t, x_b) - f_i' (t, x_b - e_i \delta t) \right] e_i
\]

(23)

For further details, the readers are referred to [41] where several validation cases are presented for this new implementation based on the coordinate transformation. The above transformation method can be incorporated with any interpolated bounce-back scheme. In the present simulation, this new bounce back scheme is implemented with the linear interpolation scheme of [17].

3. Validation against the transient laminar pipe flow and accuracy analysis

Before discussing the LBM simulation of the turbulent pipe flow simulation, we first validate our extended MRT LBM and the new bounce back scheme
against a transient laminar pipe flow. In this case, we fix the Reynolds number $Re = \frac{u_c D}{\nu}$ at 100, where $u_c$ is the flow speed at the centerline when flow reaches the steady state. Three pipe diameters in lattice units, $D = 45$, $D = 90$ and $D = 180$ are tested in order to reveal the order of accuracy of our implementation. For each pipe diameter, we fix the shear and bulk viscosity at 0.025 and 1.0 in lattice units, respectively. The parameters that related to the MRT LBM are chosen to be identical to what listed in Table 2 except that $\lambda$ is adjusted to $-0.00648$. Two pipe wall velocities $u_w = 0$ and $u_w = -u_c$ are examined to assess the performance of our implementation with both static and moving boundaries. The flow starts from rest and is driven by a uniform body force given as $16\nu u_c / D^2$. With $D = 90$, the velocity profiles at different non-dimensional times (normalized by $D^2/(4\nu)$) for both static and moving wall cases are compared with the theoretical solutions in Fig. 4. The LBM results of the flow velocity on the Cartesian nodes are binned with 50 equally-spaced bins based on the position relative to the pipe center. As clearly indicated in Fig. 4, our implementation accurately predicts the flow velocity for both static and moving wall cases. The L2 norms of the numerical errors for different pipe diameters at several dimensionless times are calculated and presented in Table 3.
Table 3: The L2 norm and convergence rate of the numerical error of the present implementation in a laminar pipe flow simulation (top: $u_w = 0$, bottom: $u_w = -u_c$).

<table>
<thead>
<tr>
<th>$D/\delta_s$</th>
<th>$\varepsilon_{L2}(t^*=1/3)$</th>
<th>order</th>
<th>$\varepsilon_{L2}(t^*=2/3)$</th>
<th>order</th>
<th>$\varepsilon_{L2}(t^*=\infty)$</th>
<th>order</th>
</tr>
</thead>
<tbody>
<tr>
<td>45</td>
<td>6.270E-4</td>
<td>(-)</td>
<td>7.434E-4</td>
<td>(-)</td>
<td>7.841E-4</td>
<td>(-)</td>
</tr>
<tr>
<td>180</td>
<td>2.313E-5</td>
<td>1.389</td>
<td>2.728E-5</td>
<td>1.515</td>
<td>2.865E-5</td>
<td>1.567</td>
</tr>
<tr>
<td>overall</td>
<td></td>
<td>2.381</td>
<td>2.386</td>
<td>2.387</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>$D/\delta_s$</th>
<th>$\varepsilon_{L2}(t^*=1/3)$</th>
<th>order</th>
<th>$\varepsilon_{L2}(t^*=2/3)$</th>
<th>order</th>
<th>$\varepsilon_{L2}(t^*=\infty)$</th>
<th>order</th>
</tr>
</thead>
<tbody>
<tr>
<td>45</td>
<td>6.227E-4</td>
<td>(-)</td>
<td>7.547E-4</td>
<td>(-)</td>
<td>7.837E-4</td>
<td>(-)</td>
</tr>
<tr>
<td>90</td>
<td>8.474E-5</td>
<td>2.877</td>
<td>1.103E-4</td>
<td>2.775</td>
<td>1.157E-4</td>
<td>2.760</td>
</tr>
<tr>
<td>180</td>
<td>1.855E-5</td>
<td>2.192</td>
<td>2.302E-5</td>
<td>2.261</td>
<td>2.405E-5</td>
<td>2.266</td>
</tr>
<tr>
<td>overall</td>
<td></td>
<td>2.534</td>
<td>2.518</td>
<td>2.513</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The L2 norm is defined as

$$
\varepsilon_{L2} = \frac{\sqrt{\sum_x \left| u_n(x) - u_t(x) \right|^2}}{\sqrt{\sum_x \left| u_c(x) \right|^2}},
$$

where $u_n$ and $u_t$ are the streamwise velocities, respectively, from LBM and the analytical solution. The summation in the above equations are over all node points within the pipe radius. For both static and moving wall cases, Table 3 shows that the overall accuracy of the present implementation is of second order. The results based on the L1 norm yield the same conclusion.

4. Turbulent pipe flow

In this section, we compare systematically the results from our LBM simulation of turbulent pipe flow to previous results in the literature. In Table 3, we summarized the key parameters from different studies which are considered here for comparison. Although both the Reynolds number dependence and pipe length dependence are reported for the turbulent pipe flow, these dependence has a negligible impact, for the parameter ranges shown in Table 4.
Table 4: Physical and simulation parameters in the turbulent pipe flow

<table>
<thead>
<tr>
<th>Method</th>
<th>Re_r</th>
<th>N_r × N_θ × N_z</th>
<th>L_z/D</th>
<th>Resolution</th>
</tr>
</thead>
<tbody>
<tr>
<td>Eggels et al.</td>
<td>185.5</td>
<td>−</td>
<td>−</td>
<td>−</td>
</tr>
<tr>
<td>Durst et al.</td>
<td>250.0</td>
<td>−</td>
<td>−</td>
<td>−</td>
</tr>
<tr>
<td>Westerweel et al.</td>
<td>183</td>
<td>−</td>
<td>−</td>
<td>−</td>
</tr>
<tr>
<td>Eggels et al.</td>
<td>180</td>
<td>96 × 128 × 256</td>
<td>5</td>
<td>(\Delta r^+, \Delta z^+, R\Delta \theta^+) \approx (1.88, 7.03, 8.83)</td>
</tr>
<tr>
<td>uniform grid</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Loulou et al.</td>
<td>190</td>
<td>72 × 160 × 192</td>
<td>5</td>
<td>(\Delta r^+, \Delta z^+, R\Delta \theta^+) \approx (0.39 - 5.7, 9.9, 7.5)</td>
</tr>
<tr>
<td>non-uniform grid</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Wagner et al</td>
<td>180</td>
<td>70 × 240 × 486</td>
<td>5</td>
<td>(\Delta r^+, \Delta z^+, R\Delta \theta^+) \approx (0.36 - 4.32, 3.7, 4.7)</td>
</tr>
<tr>
<td>Present</td>
<td>180</td>
<td>300 × 300 × 1799</td>
<td>6.06</td>
<td>\Delta x^+ = \Delta y^+ = \Delta z^+ = 1.212</td>
</tr>
</tbody>
</table>

Only the fully-developed stage is considered in this paper. The results from the pseudo-spectral method by Loulou et al. [10] will be used as the benchmark due to the superior accuracy of the PS method. Our LBM results are first converted from Cartesian coordinate to cylindrical coordinate via volumetric binning. The detail of this process can be found in the Appendix.

4.1. Mean flow properties

We first examine the mean flow velocity profile. In Fig. 5, the curves labeled “LBM, Original” and “LBM, Modified” represent the profile with Yu’s original interpolated bounce back scheme and our modified bounce back scheme based on the same interpolation but with coordinate transformation. Due to the VGI error associated with the moving boundary, the former deviates from the other profiles significantly. However, the modified scheme effectively suppresses the VGI error leading to a profile that is in excellent agreement with those from the PS and FV methods. After addressing the VGI problem, the relative error, using the PS result of Loulou et al. as the benchmark, reduces from 4.4% to less than 1%. The remaining difference could be due to the residual VGI error, interpolation error or lattice model error on the curved boundary with the Cartesian grid. Clearly, the LDV data become inaccurate near the pipe wall.
Figure 5: The mean velocity profiles normalized by the wall scales.

A well-known fact of the mean velocity profiles at the low Reynolds numbers in a turbulent pipe is the deviation from the classical logarithmic law. Our simulation results also confirm such phenomenon.

For low-Re turbulent pipe flow, due to the insufficient separation of inner and outer scales, the mean velocity result can be normalized by either the inner scale (friction velocity, as in Fig. 5) or the outer scales (bulk flow velocity). The mean velocity profiles normalized by the bulk velocity (the respective domain-averaged flow speed and pipe radius) are shown in Fig. 6. Interestingly, under this scaling, both LBM results match the other results well, indicating that the two LBM profiles have the same shape under the outer scaling. This result confirms that the un-modified interpolation method overestimates the friction velocity, which is an outcome of the VGI error contaminating the wall shear stress.

4.2. Basic turbulence statistics

The Reynolds stress profiles are compared in Fig. 7. The straight line indicates the total stress $-u_z^+ u_r^+ + dU_z^+ / dr^+$, which can be obtained from the
momentum balance equation in the streamwise direction. As shown in Fig. 7, both LBM results are in excellent agreement with the other DNS results, implying the VGI error does not have an obvious impact on the turbulent fluctuations. This makes sense because the turbulent stress measures the level of velocity fluctuation which is independent from the frame of reference. The two experimental results, however, exhibit significant differences from the DNS counterparts, especially in the near wall region, this could be due to the difficulty of conducting accurate measurements in the near-wall region and other measurement errors. This is perhaps one obvious example where experimental data should not be used as the benchmark.

Root mean squared (r.m.s.) fluctuation velocity profiles in the streamwise, radial, and azimuthal directions are shown in Fig. 8, Fig. 9, and Fig. 10, respectively. Again, the VGI error only introduces a minor effect on the r.m.s velocity profiles. Only in the buffer region where the peaks in these profiles occur, and the pipe center region, small visible differences can be observed. The peaks in the streamwise, radial, and azimuthal velocity fluctuations are located at 15.3,
The Reynolds stress profiles as a function of radial location.

Figure 7: The Reynolds stress profiles as a function of radial location.

56.7, 37.8, respectively, in LBM, compared to 14.7, 57.2, 36.3, respectively, in PS. The modified boundary treatment always leads to slightly smaller velocity fluctuations compared to the original treatment, which yields a slightly better match with the PS results in the buffer region but slightly worse agreement near the pipe center.

It is worth noting that the FV profiles of [9] fail to match the other results, especially for the radial and tangential components, with relative errors in the range of 10% to 15% when compared to the results of [10], indicating clearly that Eggels’ FV data is inaccurate. This could be mainly due to their use of a uniform grid in the radial direction causing the near wall region to be inadequately resolved (see Table 4). The grid resolution in our LBM simulations is $\delta x^+ = \delta y^+ = \delta z^+ = 1.212$, compared to $\delta r^+ = 1.88$ in [9]. It has been suggested that the minimum resolution should be $\delta x^+ = \delta y^+ = \delta z^+ = 2.25$ for DNS of a turbulent channel flow [3, 8]. The curved boundary in the turbulent pipe could impose a more demanding grid-resolution requirement. Furthermore, since the advection term in LBM is treated exactly (except at the pipe wall
where the interpolated bounce back is introduced), leading to a much smaller numerical dissipation in LBM when compared to the second-order FV method used by Eggels et al. [9].

It is interesting to note that while our LBM simulation based on D3Q19 lattice grid presents highly accurate mean and r.m.s velocity results compared to PS results, significant (up to 30%) deviations were reported in previous LES study [6]. Such deviations were attributed to the lack of isotropy of D3Q19 in representing circular pipe boundary, since the results from the simulation based on D3Q27 lattice that has better isotropy showed good agreement with the FV results by Eggels et al. [9]. Based on our comparisons above, we conclude that the FV simulation on uniform grid by Eggels et al. [9] should not be used as the benchmark. Also, it is more likely that the large deviations reported in the previous LES study [6] are partially due to the insufficient resolution (the highest resolution in that study was $\delta x^+ = \delta y^+ = \delta z^+ = 3.6$) for the near wall turbulence. Due to the lack of grid resolution, the general good results based on the D3Q27 lattice grid in [6] are also questionable.
Figure 9: Profiles of the r.m.s. velocity fluctuations in the radial direction.

Figure 10: Profiles of the r.m.s. velocity fluctuations in the azimuthal direction.
According to the radial momentum balance equation, the mean pressure can be related to the r.m.s. velocities in the radial and azimuthal directions as

\[ \frac{1}{\rho} P_r(r) + \overline{u_r^2}(r) + \int_r^R \frac{\overline{u_\theta^2}(r) - \overline{u_r^2}(r)}{r} dr = \frac{1}{\rho} P(R) = \text{const.} \]  

(25)

This balance is well captured by our LBM results for most of the region as shown in Fig. 11, where \( \langle P \rangle, \Phi \) and \( \zeta \) correspond to the first, second and third terms on the LHS of Eq. (25), respectively. It is worth emphasizing that LBM solves the weakly compressible N-S equations with a divergence term that on the order of \( O(Ma^2) \). Compared with the fully incompressible N-S equations, the *incompressible-equivalent* definition of the pressure from LBM should be

\[ P = \tilde{P} + \rho \left[ \frac{2}{3} \nu (\nabla \cdot \vec{u}) - \nu V (\nabla \cdot \vec{u}) \right] \]  

(26)

where \( \tilde{P} = c_s^2 \delta \rho \), and \( P \) is the corrected pressure that resembles the incompressible flow. In our simulation, due to the use of the large bulk viscosity to enhance numerical stability, the error caused by the third term on RHS of Eq. (26) is significant and must be removed using the above pressure correction in order to satisfy Eq. (25). Very close to the pipe wall, the LBM pressure profile exhibits small fluctuations and slight deviation of the sum from the constant in the region \( r/R > 0.92 \). This results from two factors: the pressure noise associated with the moving boundary in LBM (as demonstrated later), and the numerical error when calculating \( \zeta \). In the post-processing, we computed this integral by trapezoidal rule, which may not be accurate since the r.m.s velocity in both radial and tangential region are changing rapidly near the wall.

The r.m.s. pressure profiles are presented in Fig. 12. For the LBM result, the sudden jump near wall is mainly due to noises associated with the acoustic waves at the moving curved boundary, as indicated by an instantaneous snapshot of the pressure contour at a random cross section (here \( z = (N_z - 1)/2 \)) in Fig. 13. Other than that, the r.m.s pressure profile of LBM is in better agreement with the PS profile than the FV results of [9]. Our result is also slightly better than that of [11] near the peak of r.m.s. pressure. Specifically, the FV results of [9] contain roughly 10% relative error when compared with the PS result of [10].

29
Although it is well known that LBM only has a first-order accuracy for the pressure calculation, the results here show that LBM can provide better results on the simulated velocity and shear stress over the second-order FV method. The latter method solves the pressure Poisson equation to update the pressure field.

Next, the averaged dissipation rate profile is presented in Fig. 14. The minor oscillation for $\delta^+ < 3$ in the LBM result is mainly due to the pressure noise and binning error near the wall. The LBM result is in very good agreement with the PS result of [10], and the FV result of [11] based on a non-uniform grid.

Finally, the r.m.s. vorticity profiles in the streamwise, radial, and azimuthal directions are exhibited in Fig. 15. In LBM, the vorticity is calculated in the Cartesian coordinates based on a second-order central finite-difference approximation and then projected to the cylindrical coordinates. For the boundary nodes, special treatment is necessary to maintain the same accuracy of vorticity calculation in the whole field when straight forward central differencing does not apply (see the Appendix). As indicated in Fig. 15 in general the r.m.s.
Figure 12: Profiles of the r.m.s. pressure fluctuations.

Figure 13: A instantaneous snapshot of pressure contours at a random cross section (here $z = (N_x - 1)/2$)
vorticity profiles from LBM are in better agreements with the PS benchmark results than their FV counterparts. In terms of the high-order statistics, LBM in the Cartesian coordinates could have better performance than the FV with a uniform grid in cylindrical coordinates. In Fig. 15 we also present the r.m.s. vorticity profiles from the classical PS DNS of turbulent channel flow [44]. The very similar profiles imply the similar vortical structures in the two flows.

4.3. High-order statistics in physical space

The skewness and flatness profiles of the velocity fluctuation in each direction are also calculated. The skewness $S(u'_{\alpha})$ and flatness $F(u'_{\alpha})$ are defined as

$$S(u'_{\alpha}) = \frac{u'^{3}_{\alpha}}{u'^{2}_{\alpha}}^{3/2}, \quad F(u'_{\alpha}) = \frac{u'^{4}_{\alpha}}{u'^{2}_{\alpha}}.$$  

(27)

As shown in Fig. 16 for streamwise velocity fluctuation, the LBM skewness profiles match well with the PS and LDA data, especially for the near wall region. For the radial velocity fluctuation, the LBM result shares similar qualitative trend of reduced skewness as the PS and LDA results near the wall, but the LBM results have an unphysical jump right at the pipe wall. Such
Figure 15: Profiles of the r.m.s. vorticity fluctuations: (a) streamwise vorticity, (b) radial vorticity, (c) tangential vorticity.
unphysical jump is likely associated with the acoustic waves generated due the moving pipe wall. In the azimuthal direction, the skewness should be identically zero since the flow has no preference in the azimuthal direction. In that sense, our LBM results could be even more trustworthy than the PS result as the PS skewness in the azimuthal direction oscillates more significantly from zero. This could be a result of insufficient time duration to average out the deviations.

According to [10], the PS statistics are averaged over 46 fields at different times spanning 5.82 large eddy turnover times (defined as $R/u_\tau$). In our simulation, a much longer period (with 2300 snapshots covering 60.1 large eddy turnover times) was used. Due to the stronger fluctuations of high order statistics, it usually requires a longer averaging period to reduce the uncertainty. The LBM flatness profiles are also in good agreements with the PS benchmark data, except for a thin layer very close to the wall. Both the flatness of streamwise and radial velocity in this region exhibit a small jump, again likely a result of contamination by the acoustic waves near the moving pipe wall.

4.4. Statistics in spectral space and analyses of flow length scales

To better quantify the flow scales in the turbulent pipe flow, we calculate the one-dimensional auto-correlations functions and energy spectra for all velocity and vorticity components at three selected radial locations $\delta^+ \approx 3.5$, $\delta^+ \approx 22.0$ and $\delta^+ \approx 122.0$, which correspond to the viscous sublayer, buffer region and logarithmic region, respectively. The auto-correlation functions are calculated in physical space, as

$$Q_{\alpha\alpha} (\delta_z, r) = \frac{u'_\alpha (r, z, \theta) u'_\alpha (r, z + \delta_z, \theta)}{u_{\alpha,rms}^2(r)}, \quad (28a)$$

$$Q_{\alpha\alpha} (\delta_\theta, r) = \frac{u'_\alpha (r, z, \theta) u'_\alpha (r, z, \theta + \delta\theta)}{u_{\alpha,rms}^2(r)}, \quad (28b)$$
Figure 16: The skewness profiles of velocity components: (a) streamwise velocity, (b) radial velocity, and (c) azimuthal velocity.
Figure 17: The flatness profiles of velocity components: (a) streamwise velocity, (b) radial velocity, and (c) azimuthal velocity.
where \( \langle \ldots \rangle \) denotes averaging over the azimuthal (or streamwise) direction and over time. The streamwise and azimuthal energy spectra are given as

\[
\begin{align*}
E_\alpha(k_z; r) &= \frac{1}{2} \sum_{k_\theta} [\hat{u}_\alpha(r, k_\theta, k_z) \hat{u}_\alpha^*(r, k_\theta, k_z) + \hat{u}_\alpha(r, k_\theta, -k_z) \hat{u}_\alpha^*(r, k_\theta, -k_z)] \\
E_\alpha(k_\theta; r) &= \frac{1}{2} \sum_{k_z} [\hat{u}_\alpha(r, k_\theta, k_z) \hat{u}_\alpha^*(r, k_\theta, k_z) + \hat{u}_\alpha(r, -k_\theta, k_z) \hat{u}_\alpha^*(r, -k_\theta, k_z)]
\end{align*}
\]  

(29a)

(29b)

where \( \hat{u}_\alpha(r, k_\theta, k_z) \) is the velocity or vorticity component in the Fourier space, which is defined as

\[
\hat{u}_\alpha(r, k_\theta, k_z) = \frac{1}{N_z N_\theta} \sum_j \sum_k u_\alpha(z_k, \theta_j, r) \exp \left( -i \frac{2\pi k_z \cdot z_k}{L_z} - ik_\theta \cdot \theta_j \right)
\]  

(30)

and \( \hat{u}_\alpha^*(r, k_\theta, k_z) \) is its complex conjugate.

As shown in Fig. 18, for the logarithmic region (Fig. 18(c)), the LBM results of two-point auto-correlation functions of azimuthal and radial velocity are in good agreement with corresponding PS results. However, for the two near wall locations (Fig. 18(a) and 18(b)), significant difference are observed for the streamwise velocity component between the PS and LBM correlation functions, as the latter decay much slower than the former. This perhaps indicates that the near-wall radial resolution in our LBM simulation needs to be further refined.

It is also interesting to note that the PS auto-correlation functions for three velocity components are slightly negative, even at the largest separation distance \( \delta_z = 2.5D \). Such behavior makes the PS results questionable since at the largest separation, the velocity components should be uncorrelated if the pipe length is sufficient to eliminate the effect of periodic boundary condition in the streamwise direction. The same issue also exists in the FV correlation function of streamwise velocity component. Chin et al. [13] investigated the sufficient pipe length for converged turbulence statistics at different radial locations. Their result (Fig. 4a in [13]) suggests for \( Re_\tau = 170 \), the smallest separation distance for the streamwise velocity to decorrelate is roughly \( \delta_z^+ = 1200 \), which corresponds to a sufficient pipe length of 6.6\( D \). In that sense, neither our LBM or
PS results at large separation distances can be viewed as perfect. However, negative velocity correlation has not been reported in Chin et al. in the near-wall region, which may imply that the PS results in this case is inaccurate.

For the vorticity components, the spatial correlations are shown in Fig. 19. Although all results are in good qualitative agreements, significant quantitative differences can be observed between the PS correlation functions and their LBM counterparts, especially in the near-wall region. Again, such differences could either due to the insufficient near wall grid resolution in our LBM simulation, or the insufficient pipe length in the both PS and our LBM simulations. Away from the wall, the vortex structures are more isotropic, as the correlation functions of three vorticity components essentially overlap.

The azimuthal correlation functions from LBM and PS are shown in Fig. 20 and Fig. 21 for velocity and vorticity, respectively. In these cases, the FV data are not available, and the PS data are only available for the near-wall and buffer regions. Here, the PS correlations for velocity vorticity components do decay to zero as the azimuthal separation is increased. In general, the LBM results and PS results are in good agreement.

The one-dimensional energy spectra in both streamwise and azimuthal directions for all velocity and vorticity components are presented in Fig. 22 to Fig. 25 at the same selected radial locations. In order to compare with the benchmark results, we rescaled the PS benchmark dataset by multiplying a factor \( \sum_{k_z \text{ or } k_\theta} E_{\alpha,LBM} / \sum_{k_z \text{ or } k_\theta} E_{\alpha,PS} \) such that the areas under the pair are the same. In general, both streamwise and azimuthal energy spectra obtained from the LBM simulations are in excellent agreement with their PS benchmark curves for all velocity and vorticity components at the three different locations. The deviations in the streamwise spectra in the logarithmic region may be related to the smaller number of data points near the pipe center, while the deviations in the azimuthal spectra at large wavenumbers are likely due to the insufficient number of bins in the azimuthal direction used in data conversion from the Cartesian to cylindrical coordinates. The energy spectra for both velocity and vorticity components confirm the nearly isotropic flow in the pipe center. It is
Figure 18: The streamwise two-point auto-correlation functions for velocity components at different locations, (a): near wall ($\delta^+ = 3.5$), (b): buffer region ($\delta^+ = 22.0$), (c): logarithmic region ($\delta^+ = 122.0$). FV results [9] are taken at slight different representative radial locations, as $\delta^+ \approx 4$, $\delta^+ \approx 17$ and $\delta^+ \approx 91$ in (a), (b) and (c), respectively.
Figure 19: The streamwise two-point auto-correlation functions for vorticity components at different locations, (a): near wall ($\delta^+ = 3.5$), (b): buffer region ($\delta^+ = 22.0$), (c): logarithmic region ($\delta^+ = 122.0$).
Figure 20: The azimuthal two-point auto-correlation functions for velocity components at different locations, (a): near wall ($\delta^+ = 3.5$), (b): buffer region ($\delta^+ = 22.0$), (c): logarithmic region ($\delta^+ = 122.0$).
Figure 21: The azimuthal two-point auto-correlation functions for vorticity components at different locations, (a): near wall ($\delta^+ = 3.5$), (b): buffer region ($\delta^+ = 22.0$), (c): logarithmic region ($\delta^+ = 122.0$).
particularly interesting to observe an unphysical peak in the azimuthal spectrum of the azimuthal velocity (Fig. 24(a)) at $k_{\theta} = 4$ in the near wall region. This could be a result of the weak unphysical secondary flow pattern caused by the use of D3Q19 lattice, as reported in [19, 6, 18]. The azimuthal spectra of radial and streamwise vorticity components (Fig. 25(a)) also exhibit the similar weak unphysical behavior at $k_{\theta} = 4$, which corresponds to the unphysical pattern in the azimuthal spectrum of azimuthal velocity. However, compared with the significant secondary flow that reported in the previous LES study [6], the secondary flow observed in the present simulation is much weaker. In Fig. 26 we show the cross-sectional contour of the averaged streamwise velocity over about 44.6 large eddy turnover times. Although a weak secondary flow patterns can be still observed, the contours generally have no obvious azimuthal dependence. The much weaker secondary flow due to numerical artifact in our LBM is due to several factors: first, we shift the pipe center a little bit so the lattice arrangement is no longer perfectly symmetric; second, compared with [6] who used only 100 lattice units for the pipe diameter, our resolution is three times better; finally, our improved boundary implementation also likely reduces such artifact.

5. Summary and conclusions

In this work, we present a direct numerical simulation of a fully developed turbulent pipe flow at the bulk flow Reynolds number $Re_b \approx 5200$ (or $Re_\tau = 180$), using the lattice Boltzmann method on a D3Q19 lattice. A number of difficulties associated with accuracy and numerical instability of LBM involving a curved wall have been resolved to make this possible for the first time. To handle this turbulent flow with reasonable computational resources, an extended MRT LBM model has been introduced to improve the numerical stability of the simulation by relaxing the strict relationship between the viscosities and relaxation parameters in standard LBM. To accelerate the transition from laminar to turbulence, a divergence free perturbation force field is introduced
Figure 22: The streamwise energy spectra for velocity components at different locations, (a): near wall ($\delta^+ = 3.5$), (b): buffer region ($\delta^+ = 22.0$), (c): logarithmic region ($\delta^+ = 122.0$). FV results [9] are taken at slightly different representative radial locations, as $\delta^+ \approx 4$, $\delta^+ \approx 17$ and $\delta^+ \approx 91$ in (a), (b) and (c), respectively.

Figure 23: The streamwise energy spectra for vorticity components at different locations, (a): near wall ($\delta^+ = 3.5$), (b): buffer region ($\delta^+ = 22.0$), (c): logarithmic region ($\delta^+ = 122.0$).
Figure 24: The azimuthal energy spectra for velocity components at different locations, (a): near wall ($\delta^+ = 3.5$), (b): buffer region ($\delta^+ = 22.0$), (c): logarithmic region ($\delta^+ = 122.0$). FV results are taken at slightly different representative radial locations, as $\delta^+ \approx 4$, $\delta^+ \approx 17$ in (a) and (b), respectively.

Figure 25: The azimuthal energy spectra for vorticity components at different locations, (a): near wall ($\delta^+ = 3.5$), (b): buffer region ($\delta^+ = 22.0$), (c): logarithmic region ($\delta^+ = 122.0$).
Figure 26: Contours for averaged streamwise velocity. The averaging is taken over the streamwise direction and over time. Here a duration of 44.6 eddy turnover times was used.
to excite the flow field at the early stages. The simulation set-up and other implementation details have also been presented. Compared with the turbulent channel flow simulation, we discovered a more severe $Ma$ constraint due to the presence of a strong checkerboard instability. To overcome this problem, we added a negative velocity to the pipe wall to reduce the velocity magnitude in the center region of the pipe. This novel treatment however, brings another issue related to the Galilean invariance on the pipe wall surface. This issue is resolved by an improved bounce back scheme based on coordinate transformation. We believe these implementation details in our work provide a useful guidance for others to conduct similar simulations in the future.

Next, the results from the LBM simulations are compared systematically with previous DNS and measurement data. We have demonstrated that the violation of Galilean invariance (VGI) error in the usual interpolated bounce back causes a significant error on the mean velocity, although this seems to have a minor impact on the fluctuating velocity statistics. The VGI error on the mean velocity can be removed by using the improved bounce back based on coordinate transformation. The comparisons in the r.m.s. velocities, using the spectral simulation data [10] as the benchmark, reveal that the LBM results are much more accurate than the second-order finite difference results on a uniform grid. The deficiency of the FV results is likely due to the inadequate grid resolution near the pipe wall, implying that DNS of the turbulent pipe flow requires a more demanding grid resolution requirement (say $\delta r^+ \leq 1.25$) than the case of the turbulent channel flow ($\delta \leq 2.25$) at similar friction Reynolds numbers. Additionally, LBM treats the advection in the mesoscopic space exactly (except when the interpolated bounce-back is used near the curved wall) and preserves the exact local mass and momentum conservations, implying that LBM could be physically more accurate than the second-order FV method based on solving the Navier-Stokes equations. The r.m.s. pressure and vorticity comparisons indicate that the LBM has a better accuracy than the second-order FV methods on both uniform and non-uniform grids for the velocity gradient calculations. The comparisons of higher-order statistics (skewness, flatness) of velocity fluc-
tations again confirm the better accuracy of the LBM than the second-order FV on uniform grid. Therefore, we argue that the early DNS data by Eggels et al. are rather inaccurate and as such should not be used as a benchmark in the future. Previously, the DNS data by Eggels et al. have often been used as a benchmark, for example, in [6], [18] and [45].

These comparisons also reveal a problem in LBM related to pressure fluctuations and high-order velocity statistics near the wall (within roughly 3 grid spacings from the wall or the last two bins near the wall with a bin width of 1.8 in wall units). This indicates that the novel LBM implementation at the wall discussed in this paper is by no mean perfect. Fortunately, this problem in LBM is local and does not contaminate much the solution away from the wall. In a few cases such as the simulated spatial correlations and high-order statistics away from the wall, due to either longer simulation time and finer spatial resolution in the streamwise direction, the LBM data appear to be more accurate than the spectral simulation results.

A weak unphysical energy peak at \( k_\theta = 4 \) appears in the azimuthal energy spectra of tangential velocity, radial and streamwise vorticities, which could be caused by the unphysical secondary flow related to the use of D3Q19, aka, the lattice effect [19] [6] [18]. However, we demonstrate that this artifact in our LBM is much weaker than those found in [6], namely, the mean velocity profile in our LBM deviates from the spectral benchmark by less than 1%, compared to 28% in [6]. This dramatic improvement is related to several reasons: the shifting of lattice nodes relative to the pipe center, use of fine grid resolution, and an improved implementation of boundary condition.

6. acknowledgments

This work has been supported by the U.S. National Science Foundation (NSF) under grants CNS1513031, CBET-1235974, and AGS-1139743 and by Air Force Office of Scientific Research under grant FA9550-13-1-0213. Computing resources are provided by National Center for Atmospheric Research through
7. Appendix

7.1. Data post-processing details

All results in our turbulent pipe flow simulation are presented in cylindrical coordinates. The LBM results on the Cartesian grid are first transformed into cylindrical coordinates through the following binning and projection:

1. Divide the pipe cross section into 100 equal spaced bins according to their radial locations and number the bins as bin 1, 2, ..., 100 from pipe center to the pipe wall. Since the pipe radius is 148.5, the bin width is 1.485 lattice units. Each lattice node is associated with a square lattice cell of width equal to one lattice unit and the node at the center. Clearly, a given lattice cell can at most overlaps partially with two bins since the cell diagonal length \( \sqrt{2} = 1.414 \) is less than 1.485, as illustrated in Fig. 27.

2. For each square lattice cell (3D cubic lattice cell projected in 2D), identify the corner points \( a_0 \) and \( a_1 \) corresponding to the shortest and longest distances (\( d_0 \) and \( d_1 \)) from the pipe centerline, respectively.

3. Find out in which bin/bins the points \( a_0 \) and \( a_1 \) are located, then one of the following cases must apply:

   (a) Both \( a_0 \) and \( a_1 \) are located in the same bin \( i \) (Fig. 27, Case 1), then add all the contribution (i.e., \( p(i) = 1 \)) from this particular cell into bin \( i \).

   (b) \( a_0 \) is located in bin \( i \) and \( a_1 \) is located in bin \( i + 1 \) (Fig. 27, Case 2), then the contribution from this lattice cell is partitioned into Bin \( i \) and Bin \( i + 1 \), according to the following relative percentages \( p(i) \) and \( p(i + 1) \), respectively,

\[
p(i) = \frac{l_{i,i+1} - d_0}{d_1 - d_0}, \quad p(i + 1) = \frac{d_1 - l_{i,i+1}}{d_1 - d_0}, \quad (31)
\]
where $l_{i,i+1}$ is the radial location of the boundary between bin $i$ and bin $i+1$. While the above is not based on the precise overlap area of the cell and a bin, but test calculations show that this partition provides a sufficiently accurate approximation compared to the true area.

(c) $a_0$ is located in the last bin (Bin 100) and $a_1$ is located outside the pipe wall (Fig. 27, Case 3), then define $p(i)$ ($i = 100$ in this case) as

$$p(i) = \frac{R - d_0}{d_1 - d_0}. \quad (32)$$

Each quantity is multiplied by the $p$ value defined above, before the contribution to the bin from a particle cell is added. The average value for the bin is computed by the respective sum over all fluid cells divided by the bin volume. This post-processing strategy is applied to calculate statistical profiles. In addition to these profiles, two-dimensional binning on cylindrical surfaces of different radial locations are also presented to study the azimuthal correlation and spectra in Sec. 4.4. In azimuthal direction, each bin is evenly divided into 100 cells based on the azimuthal location of the nodes. Unlike in the radial direction where the above volumetric projection is applied, since in the azimuthal direction the flow is expected to be homogeneous, the information at each Cartesian
7.2. Vorticity calculation

Unlike the strain rate components, the vorticity calculation in LBM cannot be done in the particle distribution space, but has to rely on finite difference approximations, when D3Q19 lattice is used [46]. In this work, the vorticity components are calculated first in the Cartesian coordinates, and then transformed into the cylindrical coordinates. A second-order central difference scheme is applied to calculate each velocity-gradient component necessary for the calculation of vorticity components. For boundary nodes, the wall velocity is used in the situations where necessary node points for central differencing are not available. Taking \( \frac{\partial u_y}{\partial x} \) as an example, there are four different situations in the vorticity calculation, as shown in (a), (b), (c) and (d) in Fig. 28. For each case, the vorticity at node \((i, j, k)\) is calculated as follows

1. Case (a),
\[
\frac{\partial u_y}{\partial x} (i, j, k) = \frac{u_y (i + 1, j, k) - u_y (i - 1, j, k)}{2\Delta x}
\] (33)
2. Case (b),
\[ \frac{\partial u_y}{\partial x} (i, j, k) = \frac{q^2 u_y (i + 1, j, k) - u_{wy} + (1 - q^2) u_y (i, j, k)}{(q^2 + q) \Delta x} \] (34)

3. Case (c),
\[ \frac{\partial u_y}{\partial x} (i, j, k) = -\frac{q^2 u_y (i - 1, j, k) - u_{wy} + (1 - q^2) u_y (i, j, k)}{(q^2 + q) \Delta x} \] (35)

4. Case (d),
\[ \frac{\partial u_y}{\partial x} (i, j, k) = \frac{(q_1 - q_2) u_{wy} - u_y (i, j, k)}{q_1 q_2} \] (36)

References


