INCORPORATING TURBULENCE MODELS INTO
THE LATTICE-BOLTZMANN METHOD*

CHRISTOPHER M. TEIXEIRA
Exa Corporation, 450 Bedford Street
Lexington, Massachusetts 02420, USA
E-mail: teixeira@exa.com

Accepted 28 September 1998

The Lattice-Boltzmann method (LBM) is extended to allow incorporation of traditional
turbulence models. Implementation of a two-layer mixing-length algebraic model and
two versions of the $k - \epsilon$ two-equation model, Standard and RNG, in conjunction with
a wall model, are presented. Validation studies are done for turbulent flows in a straight
pipe at three Re numbers and over a backwards facing step of expansion ratio 1.5 and
Re$_H$ = 44 000. All models produce good agreement with experiment for the straight pipes
but the RNG $k - \epsilon$ model is best able to capture both the recirculation length, within
2% of experiment, and the detailed structure of the mean fluid flow for the backwards
facing step.

Keywords: Lattice-Boltzmann Method; Turbulence Models; Validation; Backwards-
Facing Step.

1. Introduction

The primary motivation, from a physics perspective, for pursuing new methods to
simulating fluid dynamics that use a discrete, pseudo-microscopic approach, such as
Lattice-Gas Automata (LGA)$^1$ or the Lattice-Boltzmann method (LBM)$^2$ and its
derivatives, is that the dynamics are simpler and more general than the continuum
approach (i.e., Navier–Stokes equations). By simplifying the underlying physics so
that only the key elements (conservation laws, symmetry conditions, proper equi-
librium distributions) needed to assure accurate macroscopic behavior are retained,
a computational advantage over traditional methods can be achieved,$^2$ especially
when the inherently parallel nature of such methods is exploited.

Consequently, the LGA and LB methods have found application in many areas
of flow physics$^3$ including both directly simulated and modeled single-phase turbu-
 lent flows, flows in complex geometries, multiphase and multicomponent flows, and
flows undergoing heat transfer and chemical reactions. In this work, we focus on

*This paper was presented at the 7th Int. Conf. on the Discrete Simulation of Fluids held at the
further extending the turbulence simulation capability of the method by describing how standard eddy-viscosity approaches such as algebraic and two-equation models (specifically the original and RNG $k - \epsilon$ models) can be incorporated into it. We also discuss the requirements for the application of the Large-Eddy Simulation (LES) technique. These models are used in conjunction with a wall model that has been developed to enforce a shear stress consistent with the law-of-the-wall for turbulent boundary layers independent of the orientation of the wall with respect to the underlying grid. This permits the method to simulate high Reynolds number complex flows without having to pay the computational expense of resolving down to the wall. The turbulence models, in conjunction with the lattice method, are applied to standard turbulent flow benchmarks such as flow in a straight pipe and flow over a backwards facing step.

2. Overview of Fluid Simulation Method

The fluid simulation technique used was a hybrid of an extended lattice-gas method called Digital Physics$^4$ and the Lattice-Boltzmann BGK approach.$^{5,6}$ Here, we provide only a cursory overview of the method and leave the details to the indicated references.

The primary computational element is the particle occupation, also known as the distribution function, in discrete phase space (space, velocity, time and energy are all discretized). This quantity is denoted $N_{ji}(x,t)$, where $j$ is the energy of a state moving in discrete direction $i = (1,2,\ldots,d_j)$. The velocity of such a state is $c_{ji}$. Particles, with unit mass, have energy $j = \frac{1}{2}(c_{ji} \cdot c_{ji})$. Either floating point or fixed point (i.e., integers$^a$) representation of the particle occupation can be used. The velocity and energy discretization is dictated by symmetry requirements. Traditionally, these requirements have been used to define the underlying spatial lattice which leads to a rectilinear Cartesian grid. Recently, progress has been made in separating these two discretizations leading to application of the method to non-uniform meshes.$^7$ The results presented in this work use an underlying uniform Cartesian grid only.

The microdynamics consists of two steps, advection and collision of the particle occupation values, which may be described as follows

$$N_{ji}(x + c_{ji},t + 1) = N_{ji}(x,t) + C_{ji}(\{N_{ji}(x,t)\}) ,$$

where $C_{ji}(\{N_{ji}\})$ denotes the collision operator. Here, this operator takes the single-time relaxation (BGK)$^{5,6}$ form

$$C_{ji}(\{N_{ji}(x,t)\}) = -\omega [N_{ji}(x,t) - N_{ji}^{eq}(x,t)] ,$$

The equilibrium distribution, $N_{ji}^{eq}$, is determined from statistical physics considerations in conjunction with achieving the correct macroscopic behavior;$^4$ the explicit

$^a$which allows exact enforcement of the conservation laws
form will be given shortly. The standard moments of this distribution lead to the locally conserved fluid properties mass, momentum and energy, respectively:

\[ \rho = \sum_{ji} N_{ji}^{eq}, \quad \rho u = \sum_{ji} c_{ji} N_{ji}^{eq}, \quad E = \sum_{ji} \frac{1}{2} c_{ji}^2 N_{ji}^{eq}. \]  

One of the beneficial properties of microscopic methods is that characteristics of the flow that are functions of gradients of conserved quantities, which would require nonlocal operations to compute using the continuum PDE\(^b\) approach, can be computed from higher order moments of the distribution function. Two examples of this are the stress tensor, \( \Pi = \sum_{ji} c_{ji} c_{ji} N_{ji}^{eq} \), and the energy flux, \( Q = \sum_{ji} c_{ji} \frac{1}{2} c_{ji}^2 N_{ji}^{eq} \).

Concentrating on the stress tensor, we see that this term can be split into an Euler component, \( \Pi^{(0)} \), and a dissipative component, \( \Pi^{(1)} \), which have the following form, derivable from standard Chapman–Enskog analysis\(^4,5,6\)

\[ \Pi = \Pi^{(0)} + \left( 1 - \frac{\omega}{2} \right) \Pi^{(1)} \]  
\[ \Pi^{(0)} = \sum_{ji} c_{ji} c_{ji} N_{ji}^{eq} = PI + \rho uu \]  
\[ \Pi^{(1)} \approx \sum_{ji} c_{ji} c_{ji} (N_{ji} - N_{ji}^{eq}) \]  
\[ = -2P/\omega \frac{1}{2} \left[ \frac{\partial u_\beta}{\partial x_\alpha} + \frac{\partial u_\alpha}{\partial x_\beta} - 2 \frac{D}{D} (\nabla \cdot u) I \right] \]  
\[ = -2P/\omega S, \]  

where \( P \) is the isotropic pressure, from which the temperature, \( T \), can be defined using the ideal gas equation of state, \( P = \rho T \), \( D \) is the dimension of the underlying lattice, and \( \omega \) is the single-time relaxation factor from the BGK collision operator. The expression for the kinematic viscosity, \( \nu \), is\(^4\)

\[ \nu = \left( \frac{1}{\omega} - \frac{1}{2} \right) T. \]  

The explicit functional form of the equilibrium for the system used in the simulations presented here, a three-speed model\(^8\) with \( j = (0, 1, 2) \), is as follows\(^4\)

\[ N_{ji}^{eq} = N_j^{eq}(\rho, T) \exp \left[ \frac{j}{T} \right] \exp \left[ -\frac{1}{2T} (c_{ji} - u)^2 \right] + O(u^4), \]  

where\(^8\)

\[ N_0^{eq} = \frac{\rho}{d_0} (3T^2 - 3T + 1); \quad N_1^{eq} = \frac{\rho}{d_1} 2T(2 - 3T); \quad N_2^{eq} = \frac{\rho}{d_2} T(3T - 1). \]  

\( d_1 = d_2 = 24 \) and \( d_0 \), the number of stopped particles, is arbitrary. We choose \( d_0 = 6 \). Theoretically, the method works equally well in reproducing the desired

\(^b\)Partial Differential Equation, i.e., Navier–Stokes
macroscopic hydrodynamic equations\textsuperscript{4} using either a binary sequential collision process\textsuperscript{5} or the explicit functional form,\textsuperscript{4} where the exponential is Taylor-expanded about $u = 0$ to third order in the velocity magnitude, $u$, to achieve the equilibrium distribution. In practice, to avoid instability, a very careful implementation of the direct functional form is required, the details of which will be discussed elsewhere. When this is done, the direct approach has a computational advantage over the collisional approach,\textsuperscript{9} and so this was the method used for the results to follow.

3. Turbulence Modeling

The Lattice-Boltzmann method has been used to accurately simulate fully-resolved\textsuperscript{6} fundamental turbulent flows\textsuperscript{3} at relatively low Reynolds number (Re). Unfortunately, for the high Re achieved in industrial applications, such an approach is not computationally feasible due to the wide variety of temporal and spatial physical scales excited in such problems. Instead, a model is needed for the unresolved physical scales. It is appropriate to choose an available model that is best suited to the LB method. Elements of this method that guide the choice of turbulence model are its inherent time-dependence, local nature, distribution function basis, and cheaper computational cost relative to continuum approaches. This final point means that resolution is cheaper and so more of it\textsuperscript{7} can be applied to a given problem for the same computational expense. Consequently the unresolved physics can be restricted to smaller scales, which tend to be more universal,\textsuperscript{10} and so lend themselves to simpler models.

This argument has led many researchers to investigate the LB method in conjunction with Large-Eddy Simulation (LES).\textsuperscript{11-13} While the initial work has been very encouraging, several unanswered problems within the general LES framework remain.\textsuperscript{14} These include sub-grid model determination, adaptive resolution techniques for achieving grid scale in local inertial range, and appropriate boundary condition and wall model determination. Moreover a fully three-dimensional and time-dependent simulation is required resulting in significant computational expense.

While these issues continue to be addressed, it is important to make the LB method workable with traditional, statistically averaged, approaches to turbulence modeling. Here we present implementations of algebraic and two-equation models of turbulence. The only change to the fluid algorithm is in the computation of the viscosity where, in the presence of a contribution from eddy-viscosity, $\nu_T$, the total viscosity, $\nu$, becomes a dynamic quantity

$$\nu = \left( \frac{1}{\omega} - \frac{1}{2} \right) T = \nu_T + \nu_o = \frac{1}{\omega_T} T + \left( \frac{1}{\omega_o} - \frac{1}{2} \right) T,$$  \hspace{1cm} (10)
where \( \nu_o \) represents the molecular viscosity. The local quantity \( \omega_T \) determines the eddy-viscosity and is implemented with the BGK operator, Eq. (2), using the definition,\(^\text{12} \) \( 1/\omega \equiv 1/\omega_T + 1/\omega_o \).

### 3.1. Mixing-length algebraic model

The simplest sort of algebraic turbulence model uses the Prandtl mixing length approach

\[
\nu_T = (\kappa l_{\text{mix}})^2 |S|, \tag{11}
\]

where \( |S| = (2S_{ij}S_{ij})^{1/2} \) is the magnitude of the strain rate tensor, \( S = \frac{1}{2}(\partial u_\beta/\partial x_\alpha) + (\partial u_\alpha/\partial x_\beta) \) for incompressible flows, and \( \kappa \) is the von Karman constant, \( \kappa = 0.41 \). The strain rate tensor is used to determine the turbulent time scale since it may be locally computed \(^\text{12} \) from \( \Pi^{(1)} \), Eq. (6), where now the relaxation factor \( \omega \) contains both the microscopic and eddy-viscosity contributions. With \( |S| \) defined this way, Eq. (11) is actually a quadratic equation for the unknown \( \omega_T \) \(^\text{12} \) which can be solved to give

\[
\omega_T = \frac{2\omega_o}{\sqrt{1 + 4Q\omega_o^2} - 1}, \tag{12}
\]

where \( Q = (\kappa l_{\text{mix}})^2/(2\rho T^2) \times |\Pi^{(1)}|, |\Pi^{(1)}| = (2\Pi^{(1)}_{ij}\Pi^{(1)}_{ij})^{1/2} \). The expression for \( \omega \) is \( \omega = 2\omega_o/\left(\sqrt{1 + 4Q\omega_o^2} + 1\right) \). Alternatively, \( |S| \) can be computed from \( |\Pi^{(1)}| \) using the previous time-step’s value of \( \omega \). This latter approach was used here.

A two-layer model is used to determine the mixing length. Close to the wall, the distance to the wall, \( y \), determines \( l_{\text{mix}} \) while far from the wall, this quantity is proportional to the boundary layer thickness \( l_{\text{mix}} = \min(y, \gamma \delta) \).\(^\text{(13)} \)

The problem with algebraic turbulence models is that the turbulent length scale needs to be prescribed empirically and this relation may differ from flow to flow. From an examination of experimental data for flat-plate turbulent flows, Patankar and Spalding\(^\text{15} \) suggest \( \gamma \approx 0.22 \). To use this relation for internal flows we make the fully-developed assumption that boundary layers from opposing walls reach each other so that \( \delta = R_h = \frac{1}{2}D_h \) where \( D_h \) is the local hydraulic diameter of the geometry, \( D_h = 4A_w/P_w \), in which \( A_w \) and \( P_w \) are the wetted cross-sectional area and perimeter, respectively. Thus the local mixing length determination used here becomes

\[
l_{\text{mix}} = \min(y, \varepsilon D_h), \tag{14}
\]

where \( \varepsilon = 0.1 \) and the reduced accuracy of the constant represents the approximate nature of the relation.
3.2. Two-equation models

Two-equation models for turbulence remain the most popular because they allow dynamical determination of two independent properties of the unresolved turbulence which leads to specification of characteristic velocity and length turbulent scales. The most popular of these solve dynamical equations for the turbulent kinetic energy, \( k \), and the turbulent dissipation, \( \epsilon \). Here we implement two versions of high-Reynolds number \( k-\epsilon \), the Standard model and the renormalization group (RNG) version. In these models, the Reynolds stress tensor is given by the Boussinesq approximation

\[
\tau_{ij} = 2 \mu_T S_{ij} - \frac{2}{3} \rho k \delta_{ij},
\]

where the eddy viscosity is given by

\[
\frac{\mu_T}{\rho} = \nu_T = C_\mu \frac{k^2}{\epsilon},
\]

and the equations for the turbulence quantities \( k \) and \( \epsilon \) are, respectively,

\[
\rho \frac{Dk}{Dt} = \rho \frac{\partial k}{\partial t} + \rho \mathbf{u} \cdot \nabla k = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu_o}{\sigma_{ko}} + \frac{\mu_T}{\sigma_{kT}} \right) \frac{\partial k}{\partial x_j} \right] + \tau_{ij} S_{ij} - \rho \epsilon
\]

\[
\rho \frac{D\epsilon}{Dt} = \rho \frac{\partial \epsilon}{\partial t} + \rho \mathbf{u} \cdot \nabla \epsilon = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu_o}{\sigma_{eo}} + \frac{\mu_T}{\sigma_{eT}} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{\epsilon 1} \frac{\epsilon}{k} \tau_{ij} S_{ij}
\]

\[
- \left[ C_{\epsilon 2} + f_{RNG} C_\mu \frac{\eta^3 (1 - \eta/\eta_o)}{1 + \beta \eta^3} \right] \rho \frac{\epsilon^2}{k}
\]

and \( \eta \) is a dimensionless shear rate, \( \eta = |S| k/\epsilon \). The values of the closure coefficients are, for the Standard model\( \text{superscript}^{16} \): \( C_\mu = 0.09, C_{\epsilon 1} = 1.44, C_{\epsilon 2} = 1.92, \sigma_{ko} = \sigma_{eo} = \sigma_{kT} = 1.0, \sigma_{eT} = 1.3, f_{RNG} = 0.0 \), while for the RNG model\( \text{superscript}^{17} \): \( C_\mu = 0.085, C_{\epsilon 1} = 1.42, C_{\epsilon 2} = 1.68, \sigma_{ko} = \sigma_{kT} = \sigma_{eo} = \sigma_{eT} = 0.719, f_{RNG} = 1.0 \). Finally, \( \eta_o = 4.38 \) and \( \beta = 0.012 \).

Previously,\( \text{superscript}^{11} \) it has been suggested that the \( k-\epsilon \) equations be solved within a LB structure by creating two additional populations, with components in the same directions as the particle distribution, for each of the turbulent properties and then constructing the dynamics so that these new populations solve the turbulence equations with their "macroscopic" behavior. Here, we take an alternative approach.

The only link between the turbulence equations and the LB dynamics is through the eddy-viscosity relation, Eq. (16). Consequently, we can use alternative solution methods and in fact, entirely different computational grids, to solve the two sets of equations, as long as the final eddy viscosity can be interpolated to the Cartesian grid used for the fluid simulation. In the analysis here, we apply a method developed for the transport of multiple fluid components in conjunction with the LB method\( \text{superscript}^{18} \) to the above turbulence equations.
To solve the turbulence equations, Eqs. (17) and (18), we first recast them into the following generic advection-diffusion form

$$\frac{\partial A}{\partial t} + \bar{u}_j \frac{\partial A}{\partial x_j} = D_A \frac{\partial^2 A}{\partial x_j^2} + S_A,$$  

(19)

where $A$ denotes either of the turbulence quantities, $D_A$ is the total viscosity, $D_A = \nu_0/\sigma_{Ao} + \nu_T/\sigma_{AT}$, $\bar{u}_j$ is the effective velocity, $\bar{u}_j = u_j - (\partial D_A/\partial x_j)$, and $S_A$ is the local source of quantity $A$, representing the difference between production and dissipation. This equation is then solved using an advection scheme for the scalar $A$ which is effectively a Lax–Wendroff-like finite discretization. Here, the same spatial grid is used for both the mean flow and turbulence quantities. For details, see Succi et al.\textsuperscript{18} Both implicit and explicit discretizations of the source term, $S_A$, were investigated in the pipe and backstep simulations. The explicit form was found to give acceptable results and was used in the present reported study.

To ensure realizability of the turbulence quantities, specifically that both $k \geq 0$, $\epsilon \geq 0$ and are finite, the algorithm is augmented in the following two ways. First of all, when either $k$ or $\epsilon$ is computed to be negative, the effective local viscosity is increased to the point that the problematic turbulence parameter becomes positive. This is done by reducing the negative artificial diffusion already present in the Lax–Wendroff scheme\textsuperscript{18} by recomputing the advection coefficients. By altering only the effective diffusion coefficient, conservation of the turbulence quantity due to transport can be preserved. Secondly, to ensure finite values of $k$ and $\epsilon$, whenever new values of these parameters result in the computed local eddy-viscosity becoming larger than a predetermined maximum value, $\nu_T > \nu_{T,max}$, the local turbulence quantities are not updated for that time-step. The physical reasoning behind this is that we wish to limit the change in magnitude in one of the turbulence parameters in a time-step to a fraction of its current value. Here, this fraction is about 0.1 which leads to $\nu_{T,max} \approx 0.25$ in lattice units. These two additions have made the turbulence equations solver quite robust. More sophisticated realizability schemes as well as their impact on simulation results will be the subject of further work.

This generic solution procedure can be applied to other two-equation models as well as one-equation models, such as the recent eddy-viscosity transport equation,\textsuperscript{19} derived from the $k - \epsilon$ equations, which is currently under investigation.

### 3.3. Turbulent wall model

At the large Reynolds numbers achieved in industrial, turbulent flows it is computationally undesirable to resolve all the way down to the no-slip wall. One of the standard approaches used to circumvent this requirement is to instead implement artificial boundary conditions that model the unresolved near-wall region with an estimation of the shear-stress at the wall. This can be done with the aid of the so-called universal law-of-the-wall velocity profile and empirical considerations.\textsuperscript{20,21} Instead of attempting to achieve a no-slip condition on the solid surface, the computation starts at the first cell above the surface where the fluid velocity at this
“slip” surface, \( U_s \), is typically non-zero. If the law-of-the-wall holds at this location then the fluid velocity has the well-known logarithmic profile and the shear stress at this location is the same as its value at the wall. Using standard nomenclature, the shear stress at the wall is defined in terms of the friction velocity, \( u_* \), which can be used to determine a local skin friction coefficient, \( \tau'_w \),

\[
\tau_w \equiv \rho u_*^2 = \frac{1}{2} C'_f \rho U^2
\]

and the relation between the local velocity, \( U_s \), and the friction velocity, \( u_* \), is the logarithmic relation

\[
\frac{U_s}{u_*} = \frac{1}{\kappa} \ln \left( \frac{y_s u_*}{\nu} \right) + B,
\]

where \( \kappa \) and \( B \) are empirically determined, \( \kappa = 0.41 \), \( B = 5.0 \). These relations provide two equations in the two unknowns, \( C'_f \) and \( u_* \), which can be solved in order to determine a local \( C'_f \) consistent with the law-of-the-wall. Strictly, the logarithmic profile is valid only when the dimensionless distance \( y_s^+ = y_s u_*/\nu \geq 30 \). For values of \( y_s^+ \) less than 30, profiles consistent with the buffer and viscous sub-layer regions, along with damping of the eddy-viscosity expression, Eq. (16), should be used. These near-wall extensions were not used in the results presented here.

Recently, a novel way of realizing hydrodynamic boundary conditions, based on LB dynamics, has been proposed that can be used, in particular, for the purpose of achieving the wall shear stress indicated in Eq. (20). With this approach, the mass, energy and components of momentum fluxes can be individually manipulated to achieve desired fluid dynamic behavior on an arbitrarily oriented surface. To implement this, a general three-dimensional surface is decomposed into a series of piecewise planar elements called facets. For an impermeable, adiabatic wall, the net mass and energy fluxes are zero but the momentum flux through a given facet of this surface can be constructed to achieve

\[
\text{Net Momentum Flux} = \left( p \hat{n} - \frac{1}{2} \rho U^2 C'_f \hat{u}_s \right) .
\]

where \( A \) is the surface area of this facet, \( p \) is the local fluid pressure around this facet, \( \hat{n} \) is the surface normal pointing into the fluid, \( U_s \) and \( \hat{u}_s \) are the magnitude and unit vector of the local “slip” velocity and \( C'_f \) is the locally defined skin friction coefficient, as determined by Eqs. (20) and (21). This shear stress is applied, via LB dynamics, in the direction opposed to the local velocity which naturally slows the flow down. When the local “slip” velocity is reduced to zero, the flow separates from the solid surface.

The presence of a local adverse pressure gradient serves to enhance the possibility of flow separation and so this information should be incorporated into the law-of-the-wall formulation. One approach to this is to make the assumption that the length-scale, \( y_s^+ \) in the standard log-law, Eq. (21), is simply rescaled by a scaling parameter, \( \xi_* \), that is a function of the pressure gradient. This captures the
physical consequence that the velocity profile slows down and so expands, due to the presence of pressure gradient, at least in the early stages of development. The exact form of this scaling parameter is still under investigation but for the current results, the expression

\[
\xi_o = 1 + \frac{\delta |dp|}{\tau_w ds}, \quad \left( \hat{u}_s \cdot \frac{dp}{ds} \right) > 0,
\]

\[
\xi_o = 1 \quad \text{otherwise}
\]

where \(\tau_w\) is the wall shear stress, \(dp/ds\) is the streamwise pressure gradient and \(\delta\) is a length scale of the same order of the unresolved near-wall region, was used. This approach, which gives a pressure-gradient-extended law-of-the-wall (PGE-LW) formulation is implemented by measuring \(\xi_o\) and then using \(y^+_s/\xi_o\) as the argument in the logarithm of Eq. (21).

The wall model approach can also be used to specify boundary conditions for multi-equation turbulence equations. Specifically, the boundary conditions for \(k\) and \(\epsilon\) in the law-of-the-wall region are \(25\)

\[
k = \frac{u_s^2}{\sqrt{C_k}}, \quad \epsilon = \frac{u_s^3}{\kappa y},
\]

where \(y\) is the distance from the “slip” surface to the solid wall.

4. Simulation Results

Here we present results using both the mixing-length and \(k-\epsilon\) turbulence models, in conjunction with the wall model described in the previous section, for simulations of turbulent flow in a straight pipe and over a backwards facing step.

4.1. Pipe flow

The first set of simulations were of fully-developed turbulent flow in a three-dimensional circular pipe at Reynolds numbers, \(Re_D = 40 600\) and \(424 000\), based on the bulk velocity, \(U_B\), and the pipe diameter, \(D\). This reproduces the experimental conditions of Laufer.\(^{26}\) Both simulations were performed with 50 cells across the diameter, a length-to-diameter ratio of \(L/D = 30\), and a total incoming bulk velocity of \(U_B = 0.16\) and temperature of \(T = 0.42\) in lattice units. Appropriate fully-developed turbulent boundary and initial velocity conditions for each \(Re_D\) were applied. The turbulence model implemented was the mixing-length algebraic model as given by Eqs. (11) and (14), along with the wall model.

From the initial conditions, about one complete flow-pass (< 10 000 time-steps) was needed for steady-state conditions to be achieved. Velocity profile information was then extracted from the center of the pipe. The pressure drop was measured, corrected for compressibility effects, and then used to compute a non-dimensional friction factor, \(f\). The correction for compressibility is necessary because the fluid
simulated is an ideal gas. As the pressure, $P$, decreases down the length of the pipe, so does the density, $\rho$, according to the ideal gas equation of state. Consequently, the bulk velocity increases in order to maintain continuity. Thus the measured pressure drop, $\Delta P$, has two components, a portion due to friction and a portion due to the force needed to speed up the flow to maintain continuity. If the simulation is trying to reproduce incompressible flow, or compressible flow with a much lower Mach number, this latter portion is anomalous and must be removed from the reported pressure drop, if possible. For fully-developed turbulent flows in a circular pipe, this can be done with a straightforward one-dimensional analysis of the Navier–Stokes equations\textsuperscript{27} which leads to the following expression for the non-dimensional pressure drop, $\Delta C_p$

$$\Delta C_p = \frac{\Delta P}{\frac{1}{2}\rho U_b^2} = \frac{fL/D}{1 - \frac{U_b^2}{T}}$$

so that the value of the friction factor, $f$, can be computed once the pressure drop is measured, all other parameters being given. This effect can be minimized by reducing $U_b$, or using an incompressible form of the LBM.\textsuperscript{28}

Figure 1 shows a comparison of the simulated velocity profiles with experiment for the two Reynolds numbers, nondimensionalized in terms of wall units. Care was taken to include all of the momentum in the pipe, both simulated and modeled, in the reported results. We find that the simulated profiles agree well with experiment, capturing both the logarithmic region and the defect layer for both cases. Figure 1 also contains a quantitative summary indicating that the simulated friction factor, $f$, agrees with experiment to within about 1%.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{fig1.png}
\caption{Comparison of simulated velocity profile, in wall units, with experiment for $Re = 40 600$ and 424 000. The solid line is theoretical log-law plotted for comparison purposes. As shown, the simulated friction factor, $f$, agrees with experiment to within about 1%.}
\end{figure}
from Eq. (25), differs from the experimental value of $f$, given by a correlation for smooth pipes, $f = 1.32 \ln[5.74 \text{Re}^{(-0.9)}]^{-2}$, by about 1%. While the final value of $y^+$, the location where the wall model is applied, for the lower Re case is about 50, for the higher Re it is quite large at 350. Still, as shown, this is deep inside the logarithmic region for the higher Re since this region of the flow extends from $30 \leq y^+ \leq y_{\text{max}}^+$, where $y_{\text{max}}^+ \approx 0.2 \left( \text{Re}/4 \right)^{1/2}$.

We have also simulated a two-dimensional straight pipe using the standard and RNG $k-\epsilon$ turbulence models. For this case, a lower Reynolds number was used, Re = 26 500 with $U_B = 0.1$ and $T = 0.42$. Instead of a physical length of pipe with boundary conditions, as done in the above simulations, here we use a very small simulation volume, $32 \times 32$ cells, and let the flow toroidally wrap. In order to achieve a steady-state simulation, the momentum lost to the walls at each time-step was replaced uniformly throughout the fluid via a body force (i.e., gravity). At steady-state, there is no spatial dependence in the flow direction. Besides simulations using the two versions of $k-\epsilon$ for this geometry, we also use the mixing-length algebraic model, as above, and a case with no model. Velocity profiles for the three simulations with turbulence models show the expected logarithmic and defect layer regions ($y^+ \sim 40$) while the case without a model achieves a very different, laminar-like profile, as expected (not shown). A summary of the simulated friction factor for each of the four simulations is shown in Table 1. The empirical value of $f$ at Re = 26 500 is 0.02398. Note that the compressibility correction is not needed in this case since there is no pressure drop. Instead $f$ is determined directly from its definition, $(f/8) U_B^2 \equiv u_+^2$ and measurement of $u_+$ at the wall model surface. We find that all three turbulence models agree with experimental results within 10% while the case without a model is about 40% too low. Clearly, a turbulence model is needed for this case and geometry to obtain accurate results. We plan on running the three-dimensional circular pipes of Laufer with the $k-\epsilon$ turbulence models in the future.

Table 1. Simulated values of friction factor, $f_{\text{SIM}}$, at Re = 26 500. Experimental value is 0.02398.

<table>
<thead>
<tr>
<th>CASE</th>
<th>No Model</th>
<th>Algebraic</th>
<th>$k-\epsilon$ Standard</th>
<th>$k-\epsilon$ RNG</th>
</tr>
</thead>
<tbody>
<tr>
<td>$f_{\text{SIM}}$</td>
<td>0.01429</td>
<td>0.02202</td>
<td>0.02357</td>
<td>0.02313</td>
</tr>
<tr>
<td>Error (%)</td>
<td>40</td>
<td>-8.2</td>
<td>-1.7</td>
<td>-3.5</td>
</tr>
</tbody>
</table>

4.2. **Backwards facing step**

Here we examine flow over a backwards facing step using both the algebraic and $k-\epsilon$ turbulence models. The geometry used is that of Kim et al. where the expansion ratio (ratio of outlet height to inlet height) is 1.5 and $\text{Re}_H = 44 000$ based on the inlet centerline velocity, $U_o$, and the step height, $H$. The geometry
used was two-dimensional and was 20 $H$ long beyond the step. The inlet boundary conditions were applied at 1 $H$ upstream of the step. The velocity profile used at the inlet was a turbulent boundary-layer profile that provides a boundary layer thickness at the step of $\delta_s = H/3.3$, consistent with the experiment. At the outlet, a fully-developed turbulent velocity profile was used. Using a static pressure outlet condition which allowed the velocity profile to develop self-consistently there did not produce significantly different results than presented below. Inlet and outlet profiles for $k$ and $\epsilon$ were extracted from the straight pipe data of Laufer as shown in Schetz.\textsuperscript{31}

The computations for all methods were done using two grids, both uniform, one with 30 cells along the height and the other with 50 cells. Measurements of the recirculation length changed less than 1% going from the coarser to the finer grid so the results were assumed to have converged with $H = 50$. With an incoming bulk velocity of $U_B = 0.15$, about 7,000 time-steps were required for a complete flow-pass through the system. From the same initial conditions, it was found that the algebraic model took 2.4 flow-passes to achieve steady-state, while the Standard and RNG $k - \epsilon$ models took 3.6 and 5.7 flow-passes, respectively, on the $H = 50$ grid. The longer time for the latter cases is due to the dynamic nature of the $k - \epsilon$ method.

The length of the main recirculation zone behind the step was computed for five different types of turbulence models: two algebraic models, and three using the $k - \epsilon$ model. The reported length of the experimental recirculation length, $X_R$, is\textsuperscript{30,17} $X_R/H = 7.0 \pm 0.5$ with Zhou\textsuperscript{12} et al. reporting a mean value of $X_R/H \approx 7.1$. As an aside we note that when the computation was done without an explicit turbulence model, the flow was highly unsteady with eddies being shed from the step and a mean recirculation length of $X_R/H \sim 13$; much too large. This computation is both under-resolved and two-dimensional, and so is inadequate as a direct numerical simulation of a 3D turbulent flow. Consequently, the results are poor for this case.

The five simulations are as follows: (a) two-layer mixing-length algebraic model with $D_h = 3H$ and $\epsilon = 0.1$, see Eq. (14); (b) as in (a) but with $D_h = H$ and $\epsilon = 0.25$; (c) Standard $k - \epsilon$; (d) Standard $k - \epsilon$ with pressure-gradient extended law-of-the-wall (PGE-LW), see Eq. (23); (e) RNG $k - \epsilon$ with PGE-LW. Simulations (a)--(c) use the basic law-of-the-wall wall model, Eq. (21). With the mixing-length approach it is necessary to choose a turbulent length scale. In (a) and (b) we compare results for two choices of this parameter.

Table 2 presents the computed values, along with some selected results from the literature using the traditional PDE approach for the mean fluid solver. Figure 2 shows the computed streamlines for case (e) where $X_R/H = 7.2$. We find that the algebraic model with the turbulent length scale proportional to the downstream height and the Standard $k - \epsilon$ model give the same recirculation length, $X_R/H = 6.1$, which underpredicts the experimental value by about 14%. Furthermore, Zhou et al. report the same value using the Standard $k - \epsilon$ model with a PDE fluid solver, indicating that our results are consistent with published results using the
Table 2. Computed recirculation lengths for the backwards facing step. Experimental value is $X_R/H \approx 7.1$. Top table contains current results while lower table are results from literature.

<table>
<thead>
<tr>
<th>CASE</th>
<th>(a) Alg. I</th>
<th>(b) Alg. II</th>
<th>(c) Std $k - \epsilon$</th>
<th>(d) Std $k - \epsilon$</th>
<th>(e) RNG $k - \epsilon$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$D_h = 3H$</td>
<td>$D_h = H$</td>
<td>$\epsilon = 0.1$</td>
<td>$\epsilon = 0.25$</td>
<td>with PGE-LW</td>
<td>with PGE-LW</td>
</tr>
<tr>
<td>$X_R/H$</td>
<td>6.1</td>
<td>7.3</td>
<td>6.1</td>
<td>6.3</td>
<td>7.2</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>CASE</th>
<th>Std $k - \epsilon$</th>
<th>RNG $k - \epsilon$</th>
<th>R-RNG $k - \epsilon$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$X_R/H$</td>
<td>$6.1^{32}$</td>
<td>$6.4^{32} - 6.6^{17}$</td>
<td>$6.7^{32}$</td>
</tr>
</tbody>
</table>

Fig. 2. Computed streamlines over the backwards facing step using the RNG $k - \epsilon$ turbulence model with PGE-LW wall model. Recirculation length is $X_R/H = 7.2$.

same physics model. Introduction of the pressure-gradient extended wall model into Standard $k - \epsilon$ increased the recirculation length by about 3% in the direction of experiment. The algebraic model with the turbulent length scale proportional to the step height and the RNG $k - \epsilon$ with PGE-LW produce results much closer to the experimental value at 3% and 1.5% too high, respectively. For this algebraic model, the proportionality value of $\epsilon = 0.25$ was chosen to achieve about the correct recirculation length for this case but it was later found that this same scaling gave equally good results for a backwards step of a very different expansion ratio and Reynolds number. Further investigation is ongoing and will be reported elsewhere. Improvement with the RNG $k - \epsilon$ found here is consistent with the same trend as in the literature, see the lower table in Table 2, but the magnitude of the change is much larger here. This would appear to be due to the extensions to the wall model used here but further comparisons of the data included in Zhou et al. and Yakhot et al. will bear this out. Finally, we have also included a result from Zhou et al. where a Recursion-RNG (R-RNG) model, developed therein, was used.

In order to further compare the detailed flow structure of the simulations with experiment, we provide computed results for the streamwise mean velocity profiles, streamwise turbulent intensities and the pressure profiles along the walls. Figure 3 shows the computed streamwise mean velocity profiles at six different locations beyond the step, for case (b), the algebraic model, and case (e) the RNG $k - \epsilon$ model,
Fig. 3. Streamwise velocity profiles for six locations downstream of the backwards facing step. The origin is at the step. Open circles represent the experiment; dashed line, case (b), the algebraic model; solid line, case (c), the RNG $k - \epsilon$ PGE-LW model.

Fig. 4. Streamwise turbulence intensity profiles for six locations downstream of the backwards facing step. The origin is at the step. Open circles represent the experiment; plus sign (+) are the computed results of Zhou et al. using Standard $k - \epsilon$; solid line, case (c), Standard $k - \epsilon$ with LBM; dotted line, case (e), the RNG $k - \epsilon$ PGE-LW model.

compared with experiment. These two cases provided comparable recirculation lengths that were within 5% of experiment despite using very different turbulence models. The RNG results capture the structure of the flow within the recirculation zone more accurately than the algebraic model. Both simulations recover too slowly after reattachment in comparison with the experiment.
Next, Fig. 4 shows the streamwise turbulence intensity profiles for case (c), Standard $k - \varepsilon$, and case (e) using RNG, at six locations beyond the step. To obtain values for the simulations, we assume isotropy of the turbulence fluctuations so that $uu = \frac{2}{3}k$. For the two profiles closest to the step, we compare our computed results with the computed results from Zhou et al. using the Standard $k - \varepsilon$ while for the other four, experimental results are used. Within the recirculation zone, the RNG results are smaller in magnitude than the Standard model results, while beyond the reattachment point, the computed results are very similar. Note the agreement of our Standard model results with the computed profiles of Zhou et al. Lower values of turbulence intensity within the recirculation zone indicate a lower eddy-viscosity, see Eq. (16), which results in a larger recirculation length. This is consistent with the results of Yakhot et al. Slight underprediction of the turbulence intensity shortly after the reattachment point is apparent and may be a consequence of the breaking down of the assumption of turbulence isotropy there.

Finally, we examine the coefficient of pressure, $C_p = (P - P_r)/(\frac{1}{2}\rho U_r^2)$, where $P_r$ and $U_r$ are the reference pressure and velocity taken at the centerline of the inlet, along the top and bottom walls, see Fig. 5. As in Fig. 3, we compare cases (b) and (d) of Table 1, each using the RNG $k - \varepsilon$ PGE-LW model.
Again, despite the fact that the simulations give comparable recirculation lengths, the RNG $k-\epsilon$ results are much closer to the experiment than the algebraic model. A significant difference is found in the recirculation zone along the bottom wall where the algebraic model computes a thinner and deeper $C_p$ hole than the $k-\epsilon$ computation, which agrees well with experiment.

Thus we find that the RNG $k-\epsilon$ turbulence model with PGE-LW computes a more accurate recirculation length for this backwards facing step than the Standard model using the standard values of the closure coefficients, consistent with the results of Zhou et al. and Yakhot et al., and more accurately reproduces the structure of the mean flow than an algebraic model tuned to give the desired recirculation length.

5. Concluding Remarks

It has been demonstrated that standard algebraic and two-equation turbulence models can be incorporated into the Lattice-Boltzmann method and produce results comparable to those achieved with traditional PDE fluid-flow solvers. For turbulent flow in a pipe, the friction factor for a range of Re numbers is predicted to within 10% for the algebraic model and 5% for the $k-\epsilon$ models. For the turbulent backwards facing step, better accuracy is achieved with the RNG $k-\epsilon$ model than the Standard model, with the algebraic model producing significant discrepancies in localized flow structure in the separated region, despite being able to compute an accurate reattachment length. Improvements to the wall model, such as the pressure-gradient extension discussed here, have been shown to improve the prediction of the recirculation length for this case. Of course, the validity of using a log-law type of wall model in a separated region is highly questionable but the accuracy of the pressure computations in the separated region with the RNG $k-\epsilon$ model indicate the adverse impact of this assumption may be minimal.

Because of the simplicity of the algebraic model, it requires only $\sim 5\%$ additional computational overhead as compared to Exa’s PowerFLOW™ software without a turbulence model, running on a SUN UltraSparc-2 workstation with a 167 MHz clock rate, where the update rate is about 100 000 cells per second. We do not yet have performance results for an optimized code including the two-equation turbulence models but about a 25% performance penalty, compared to the no-model code, is expected. The generic structure for solving additional PDE’s for fluid properties that interact with the main fluid presented here can be extended to solve one-equation, other two-equation, or Reynolds-stress models of turbulence.

One of the benefits of using the Lattice-Boltzmann method is the ease of which it can be applied to complex geometries. Currently, work is under way to apply the LBM with the above turbulence models to commercial internal flow applications such as HVAC ducts, intake ports and engine water jackets.

---

* Heating, Ventilation and Air-Conditioning
Acknowledgments

The author is very grateful to Sauro Succi, Hudong Chen and Kim Molvig for many useful discussions, specifically concerning achieving realizability for the turbulence parameters, and to Jamie Hoch and David Freed for suggestions on improving the performance of the code used to compute the presented results.

References