Lattice Boltzmann study on drag reduction of a bluff body by slip boundary

Liuming Yang, Yuan Gao, Shuai Zhao, Yang Yu and Guoxiang Hou *
School of Naval Architecture and Ocean Engineering, Huazhong University of Science and Technology, Wuhan, China

*Corresponding author e-mail: houguoxiang@163.com

Abstract. Slip velocity at boundary will emerge when gas film or superhydrophobic coatings are applied to the surface. The no-slip boundary conditions are no longer efficient. In this work, an improved boundary treatment is proposed to obtain slip velocity. Numerical results of the Couette flow have verified the accuracy and reliability of the improved boundary condition. Taking into account the effect of boundary slip, the present research predicts the resistance of a bluff body under low Reynolds numbers using the lattice Boltzmann method. Results show that reasonable slip distribution may be favourable to the drag reduction effect.

1. Introduction
Traditionally, it is assumed that the velocity of the liquid at the boundary equals to the velocity of the wall. It is an effective no-slip boundary condition for ordinary walls. But if the gas film or superhydrophobic coatings are applied to the surface of underwater objects, apparent slip velocity will emerge at the boundary [1]. The assumption of no-slip boundary condition is no longer efficient. Boundary slip is very meaningful to drag reduction. Farooqi et al. [2] studied the drag reduction by air injection in a pipeline and they observed a decreased pressure drop of up to 85%. Elbing et al. [3] observed a drag reduction which is larger than 80% over their tested model. Berry et al. [4] showed that significant drag reduction can be obtained by hot spheres because they have the ability to sustain a stable Leidenfrost vapor layer on the surfaces. Navier slip model is adopted for their numerical simulation. Dong et al. [5] coated a model ship with superhydrophobic surfaces and a remarkable drag reduction of 38.5% was observed. Hu et al. [6] used the alternated superhydrophobic and hydrophilic strips to trap the injected air rings on the surface of the inner rotor of the Taylor-Couette flow. The drag reduction can be up to 77.2%. However, resistance prediction of the cases with slip boundary condition is still not perfect enough and it still needs to be explored further.

The lattice Boltzmann method (LBM) is a promising tool to simulate the liquid flow with slip boundary conditions. It is crucial to reconstruct the boundary conditions. Succi [7] combined the half-way bounce back scheme and the specular reflection scheme (HBSR) to obtain slip velocity. The combination parameter is responsible for the specified slip velocity. Then Ahmed et al. [8] developed a slip boundary condition (MBSR) which is the combination of the modified bounce back scheme and the specular scheme. Wang et al. [9] related the slip length with the combination parameter theoretically to specify the slip boundary condition. But the relationship has been validated by cases of the inclined walls.
In the present work, an improved slip boundary condition is developed for 45 degrees inclined walls. The Couette flow is adopted to validate the accuracy and reliability of the improved scheme. Using the improved scheme of the lattice Boltzmann method, we predict the resistance of the bluff body. Through comparative analysis, we find that it may be helpful to the drag reduction by reasonable slip distribution.

2. Numerical method

2.1. The lattice Boltzmann method

The particle distribution function $f_i(x,t)$ is the basic quantity of the lattice Boltzmann method. It is the density of particles moving with velocity $c_i$ at position $x$ and time $t$. For the LBM, if the particle distribution functions are already known, the macroscopic density $\rho$ and velocity $u$ are calculated by

$$\rho = \sum_i f_i, \quad \rho u = \sum_i c_i f_i$$  

(1)

The most commonly two-dimensional discrete-velocity model is the standard D2Q9 model for the Navier-Stokes equation. The discrete velocities of the D2Q9 model are defined as

$$c_i = \begin{bmatrix} 0 & 1 & 0 & -1 & 0 & 1 & -1 & -1 & 1 \end{bmatrix}$$

where $c$ equals to the ratio of the lattice step $\delta_x$ to the time step $\delta_t$.

The evolution equation of the lattice Boltzmann method can be expressed as

$$f_i(x + c_i \delta_t + \delta_t, t + \delta_t) = f_i(x, t) + \Omega_i(x, t)$$  

(2)

where $\Omega_i$ is the collision operator. The Bhatnagar-Gross-Krook (BGK) operator is the commonly used operator model because it can be used for Navier-Stokes simulations and it is very simple. It is given by

$$\Omega_i(x, t) = -\frac{1}{\tau} \left[ f_i(x, t) - f_i^{eq}(x, t) \right]$$  

(3)

where $\tau$ is the dimensionless relaxation time and $f_i^{eq}(x, t)$ represents the equilibrium distribution function. Eq. 2 can be divided into two steps as below

Collision step: $\overline{f_i}(x, t) = f_i(x, t) - \frac{1}{\tau} \left[ f_i(x, t) - f_i^{eq}(x, t) \right]$  

(4)

Moving step: $f_i(x + c_i \delta_t + \delta_t) = \overline{f_i}(x, t)$  

(5)

where $\overline{f_i}$ is the distribution function after collision.

The equilibrium distribution function for D2Q9 model is expressed as

$$f_i^{eq}(x, t) = w_i \rho \left[ 1 + \frac{c_i \cdot u}{c_s^2} + \frac{(c_i \cdot u)^2}{2c_s^4} - \frac{u^2}{2c_s^2} \right]$$  

(6)

with $c_s = c/\sqrt{3}$. The weight parameters $w_i$ for D2Q9 model are given as

$$w_i = \begin{bmatrix} 4/9 & 1/9 & 1/9 & 1/9 & 1/9 & 1/36 & 1/36 & 1/36 & 1/36 \end{bmatrix}.$$  

The dimensionless relaxation time is determined by the fluid kinematic shear viscosity $\nu$ as below:
\[ \tau = \frac{V}{c_s \delta} + 0.5 \]  

(7)

2.2. **Improvement of the liquid slip boundary condition for inclined wall**

An accurate and reliable boundary condition is the key to simulate the boundary slip. In the present work, we adopt the slip boundary condition which combines the modified bounce back scheme and the specular reflection scheme (MBSR). It is given by

\[
f_i(x,t) = rf_i(x,t) + (1-r)f_i'(x,t)
\]  

(8)

where \( r \) is the combination parameter, \( -i \) is the opposite direction of \( i \) and \( i' \) is the specular reflection direction of \( i \) at the boundary.

For the liquid flows, the boundary slip is the apparent slip caused by the gas film or the air trapped in the superhydrophobic surfaces. Navier’s slip model has been employed for the liquid flow in the former researches [8, 9]. The linear slip model can be expressed by

\[
s_u = b \frac{\partial u}{\partial n}
\]  

(9)

where \( u_s \) is the slip velocity at the wall, \( b \) is the slip length and \( \partial u / \partial n \) is the velocity gradient in the normal direction of the wall.

**Figure 1.** The aligned straight wall.  
**Figure 2.** The inclined straight wall (45 degrees)

Based on the assumption of the unidirectional liquid flow near the aligned straight wall shown in Fig. 1, Wang et al. [9] deduced the relationship between the slip length and the combination parameter (Eq. 10) strictly in theory.

\[
r = \frac{\tau}{\tau + b}
\]  

(10)

But our numerical results show that this relationship is not accurate enough for the inclined wall. As shown in Fig. 2, with the assumption of the unidirectional flow near the 45 degrees inclined straight wall, we deduce the relationship of the combination parameter and the slip length.

Inspired by Wang et al. [9], the following assumptions are given

\[
\rho = \text{const}, u_{y_n} = 0, \frac{\partial \phi}{\partial x_n} = 0, \quad \frac{\partial \phi}{\partial t} = 0
\]  

(11)

where \( \phi \) represents an arbitrary variable and \( u_{y_n} \) is the velocity in the direction \( y_n \).

Based on Eq. 1, the velocities in directions \( x \) and \( y \) are obtained by
\[ \rho u_x = f_1 - f_3 + f_5 - f_6 + f_k - f_7 \]
\[ \rho u_y = f_2 - f_4 + f_5 - f_1 + f_6 - f_k \]  \hspace{1cm} (12)

So, the velocity at location \( y_n = j_n \) can be calculated by
\[ u_{j_n} = \frac{\sqrt{2}}{2} u_x + \frac{\sqrt{2}}{2} u_y \]  \hspace{1cm} (13)

The unidirectional flow suggests that
\[ f_5 - f_7 = \overline{f}_5 - \overline{f}_7 \]  \hspace{1cm} (14)

Besides, the moving law of Eq. 5 shows that
\[ f_i - f_i^1 = \overline{f}_i - \overline{f}_i^1, \quad f_i^2 - f_i^2 = \overline{f}_i - \overline{f}_i^1 \]  \hspace{1cm} (15)

where \( f_i \) means the distribution function with velocity \( \mathbf{e}_i \) at location \( y_n = j_n \).

With the above equations, the relationship of the velocities at \( j_n = 0 \) and \( j_n = 1 \) are constructed as below.
\[ u_i = u_0 + \frac{r}{r(1-r)} u_0 \]  \hspace{1cm} (16)

For the linear Couette flow, the equation can be given as
\[ u_{y_n} = \beta y_n + u_s \]  \hspace{1cm} (17)

where \( \beta \) represents the velocity gradient of the Couette flow. So the velocities at \( j_n = 0 \) and \( j_n = 1 \) are

\[ u_0 = u_s, \quad u_1 = \frac{\sqrt{2}}{2} \beta + u_s \]  \hspace{1cm} (18)

Substituting \( u_i \) and \( u_0 \) into Eq. 16, we have
\[ u_i = \frac{\sqrt{2}r(1-r)}{2r} \beta \]  \hspace{1cm} (19)

Combining Eq. 9 and Eq. 19, we can get the relationship (Eq. 10) of the slip length and the combination parameter of the MBSR scheme for 45 degrees inclined wall.
\[ r = \frac{\tau}{\tau + \sqrt{2}b} \]  \hspace{1cm} (20)

It is noted that this relationship is deduced based on the case of the linear Couette flow. And it is difficult to deduce the exact relationship between the slip length and the combination parameter for general cases. In fact, the local flow near the wall can be approximated by the linear flow if the grid is fine enough [9].
3. Numerical validation

In the above section, the relationship between the slip length and the combination parameter of the MBSR scheme is deduced strictly in theory for the 45 degrees inclined wall. In this section, we will test the improved slip boundary condition with the 45 degrees inclined Couette flow. A uniform lattice $64 \times 96$ is adopted for this case. As shown in Fig. 3, $H$ is set as $16\sqrt{2}$. The upper wall moves at a constant speed $U = 0.01$. Without special mention, all variables are in lattice units. The non-equilibrium extrapolation method is employed for the inlet, the outlet and the upper wall. The lower wall is stationary and the MBSR scheme with Eq. 20 is adopted for the slip boundary condition. Besides, we set $\tau = 1.1$ for this simulation. As shown in Fig. 4, there is nearly no difference between the numerical velocity distributions and the theoretical results. So our improved slip boundary condition is accurate and reliable enough to simulate the liquid flow near the 45 degrees inclined walls.

4. Numerical simulation and results

Researchers tried to reduce the resistance using gas film or superhydrophobic surfaces. Both gas film and superhydrophobic surfaces can produce slip velocity at the boundary. In this section, we predict the resistance of a bluff body with slip boundary condition based on the lattice Boltzmann method.

The computational domain is presented in Fig. 5. The width of the bluff body is denoted as $D$. The length of the bluff body is $2.5D$. The width of the computational domain is $15D$. We place the bluff body in the middle of the upper and the lower boundaries. The distances from the head of the bluff body to the inlet and the outlet are $7.5D$ and $22.5D$, respectively. We have the following boundary conditions.

The inlet: $u = U_0 = 0.05$, $v = 0$;

The outlet: $\frac{\partial u}{\partial x} = \frac{\partial v}{\partial x} = 0$;

The upper and the lower boundary: $\frac{\partial u}{\partial y} = 0$, $v = 0$;

The underwater bluff body: $u_n = 0$.

$u$ and $v$ represent the velocity in direction $x$ and $y$, respectively. The velocity at the inlet is denoted by $U_0$. The normal velocity at the bluff body is represented by $u_n$. The Reynolds number is defined by $Re = 2.5D \times U_0 / \nu$. The drag force $F_x$ can be obtained by momentum exchange method[10]. The drag coefficient of the bluff body is computed by $C_d = 2F_x / \rho (2.5 \bar{U}_0^2)$, where $\bar{\rho}$ is the average density of water.
In order to simulate the slip boundary, we adopt the liquid slip boundary condition Eq. 10 for edge 2 and edge 3. And the Eq. 20 is employed for edge 1. Besides, the non-equilibrium extrapolation scheme is applied to the other boundaries of the computational domain. Grid independent has been verified by simulations of $D=16, 20, 26, 30, 40$ with $Re=5, 40, 100, 150, 300, 400$. The change of drag coefficients is no more than 5% when $D \geq 20$. So, it is reasonable to set $D=30$ in the following simulations.

### 4.1. Influence of the slip length

In this subsection, the influence of the slip length on drag reduction of the bluff body is studied under Reynolds numbers $Re=5, 20, 40, 80, 150, 200, 300, 400$, respectively. The slip boundaries are applied to all the edges of the bluff body. The flows past the bluff body with slip lengths $b=0, 1, 2.5, 5, 7.5$ are simulated. The predicted drag coefficients of the bluff body are listed in Table 1.

The drag reduction rates (DR) as a function of different Reynolds number are depicted in Fig.6 for $b=1, 2.5, 5, 7.5$. From Table.1 and Fig.6, it can be noted that the drag coefficient is decreased and the drag reduction effect is enhanced with the increased slip length at the same Reynolds number. When $b=7.5$ and $5 \leq Re \leq 400$, the maximum drag reduction rate is up to 16.43% and the minimum drag reduction rate is 10.40%. It is reasonable that the drag reduction efficiency is increased with the increment of the slip length. For ideal cases, the gas film completely separates the water and the solid boundaries. The liquid-solid boundary is replaced by the gas-solid boundary. So, the slip length tends to be infinity and the drag force will be reduced drastically.

| Re  | $C_d(b = 0)$ | $C_d(b = 1)$ | $C_d(b = 2.5)$ | $C_d(b = 5)$ | $C_d(b = 7.5)$ |
|-----|--------------|--------------|----------------|--------------|----------------|
| 5   | 4.173        | 4.092        | 3.989          | 3.849        | 3.739          |
| 20  | 1.712        | 1.672        | 1.622          | 1.555        | 1.503          |
| 40  | 1.162        | 1.133        | 1.095          | 1.045        | 1.007          |
| 80  | 0.814        | 0.792        | 0.762          | 0.724        | 0.696          |
| 150 | 0.602        | 0.585        | 0.561          | 0.532        | 0.512          |
| 200 | 0.527        | 0.516        | 0.498          | 0.474        | 0.456          |
| 300 | 0.485        | 0.468        | 0.446          | 0.421        | 0.405          |
| 400 | 0.447        | 0.429        | 0.407          | 0.391        | 0.385          |
4.2. Influence of Reynolds number

Fig. 6 also shows that the drag reduction of the bluff body is influenced not only by the slip length but also the Reynolds number. We can notice that $Re=200$ is a turning point. It is because the flow changes from the steady state to the unsteady state when Reynolds number is more than 200. When $Re \leq 150$, the drag reduction efficiency becomes better with the increment of the Reynolds number for the same slip length. The same trend can be observed when $Re \geq 200$ for $b=1, 2.5$. For $b=5, 7.5$, the drag reduction rate is maximum when $Re=300$. Essentially, the slip length and the Reynolds number have great influence on the drag reduction effect because they can change the flow field.

![Figure 6. Variation of the drag reduction with the Reynolds number](image)

**Table 2. Drag coefficients of the bluff body with different slip distributions.**

| Re | Case 0 | Case 1 | Case 2 | Case 3 | Case 4 | Case 5 | Case 6 |
|----|--------|--------|--------|--------|--------|--------|--------|
| 5  | 4.173  | 3.849  | 4.119  | 3.922  | 4.172  | 3.851  | 4.112  | 3.921  |
| 20 | 1.712  | 1.555  | 1.698  | 1.583  | 1.712  | 1.555  | 1.699  | 1.583  |
| 40 | 1.162  | 1.044  | 1.156  | 1.064  | 1.163  | 1.044  | 1.157  | 1.064  |
| 80 | 0.814  | 0.724  | 0.810  | 0.737  | 0.815  | 0.723  | 0.811  | 0.734  |
| 150| 0.602  | 0.532  | 0.598  | 0.541  | 0.603  | 0.531  | 0.599  | 0.542  |
| 200| 0.527  | 0.474  | 0.526  | 0.474  | 0.531  | 0.470  | 0.528  | 0.481  |
| 300| 0.485  | 0.421  | 0.480  | 0.426  | 0.490  | 0.419  | 0.486  | 0.428  |
| 400| 0.447  | 0.391  | 0.441  | 0.391  | 0.453  | 0.388  | 0.447  | 0.391  |
4.3. Influence of the slip distribution

In the above study, the slip boundary is applied to all the edges of the bluff body. The results show that the slip boundary is beneficial to drag reduction because different slip length may bring out varied flow fields. The slip distributions may also have effect on the flow state and the drag reduction effect. As shown in Fig.5, the boundaries of the bluff body are marked as edge 1, edge 2 and edge 3. In this subsection, seven cases with different slip distributions are given as below:

Case 0: the slip boundary is applied to all edges;
Case 1: the slip boundary is only applied to edge 1;
Case 2: the slip boundary is only adopted for edge 2;
Case 3: the slip boundary is only employed for edge 3;
Case 4: the slip boundary is only used for edge 1 and edge 2;
Case 5: the slip boundary is only applied to edge 1 and edge 3;
Case 6: the slip boundary is only used for edge 2 and edge 3.

In this simulation, b=5 is set for the slip boundary. The drag coefficients for different slip distributions are presented in Table 2. The drag reduction rates of the bluff body for Case 0 to Case 6 are given in Fig.7. It is obvious that the slip distributions have great influence on the drag reduction efficiency of the bluff body. The slip boundary is not always beneficial to the drag reduction. It can be observed from the results of Case 3 that the drag resistance may be increased when the slip boundary condition is only applied to edge 3. Comparing the results of all these seven cases, we can find that the slip boundary condition of edge 2 is very crucial to reduce the drag of this bluff body. Fig.7 demonstrates that Case 2, Case 4 and Case 6 can effectively reduce the resistance of the bluff body. Case 0 employs the slip boundary for all the edges of the bluff body. The resistance reduction effect of Case 6 is even better than Case 0. So it may be a promising way to improve the drag reduction efficiency by optimized design of slip distribution.

5. Conclusion

In this research, the drag force of a bluff body with slip boundary is studied based on the lattice Boltzmann method. A modified slip boundary condition for 45 degrees inclined walls is developed through strict theoretical analysis. The simulations of 45 degrees inclined Couette flow validate the accuracy of our method.

With this slip boundary condition at hand, we study the influence of the slip length and the Reynolds number on the drag force of the bluff body. The slip boundary condition is applied to all edges of the bluff body. The drag reduction effect is enhanced with the increased slip length when \( 5 \leq \text{Re} \leq 400 \). The drag reduction efficiency is different with varied Reynolds number. \( \text{Re}=200 \) is a turning point of the drag reduction efficiency. It is because the flow state is changed when Reynolds number is near 200. Essentially, the slip length and the Reynolds number influence the drag force of the bluff body through changing the flow field jointly. To accomplish the intended drag reduction efficiency, the slip boundary should be designed carefully according to the objects.

The influence of the slip distributions is also studied. Results show that the unreasonable distributed slip boundary is not beneficial to the drag reduction. The drag resistance may be increased when the slip boundary condition is only applied to edge 3, while Case 6 even has better drag reduction effect than case with all edges employing the slip boundary. Optimal design of slip distribution may be an available method to obtain low drag force.

Acknowledgments

We acknowledge the financial support from the National Natural Science Foundation of China (Grant No.51679099 and No.51475179), and the Fundamental Research Funds for the Central Universities HUST. 2016JCTD207.

References

[1] Wang, J, Wang, B, and Chen, D. "Underwater drag reduction by gas." Friction 2.4 (2014): 295-
309.

[2] Farooqi, Sohail I., Nigel I. Heywood, and John F. Richardson. "Drag reduction by air injection for suspension flow in a horizontal pipeline." Trans. Inst. Chem. Sci. Eng 58 (1980): 16-27.

[3] Elbing, Brian R., et al. "Bubble-induced skin-friction drag reduction and the abrupt transition to air-layer drag reduction." Journal of Fluid Mechanics 612 (2008): 201-236.

[4] Berry, J. D., Vakarelski, I. U., Chan, D. Y., & Thoroddsen, S. T. (2017). “Navier slip model of drag reduction by Leidenfrost vapor layers.” Physics of Fluids, 29(10), 107104.

[5] Dong, Hongyu, et al. "Extraordinary drag-reducing effect of a superhydrophobic coating on a macroscopic model ship at high speed." Journal of Materials Chemistry A 1.19 (2013): 5886-5891.

[6] Hu, Haibao, et al. "Significant and stable drag reduction with air rings confined by alternated superhydrophobic and hydrophilic strips." Science advances 3.9 (2017): e1603288.

[7] Succi, Sauro. "Mesoscopic modeling of slip motion at fluid-solid interfaces with heterogeneous catalysis." Physical review letters 89.6 (2002): 064502.

[8] Ahmed, Nayaz Khalid, and Martin Hecht. "A boundary condition with adjustable slip length for lattice Boltzmann simulations." Journal of Statistical Mechanics: Theory and Experiment 2009. 09 (2009): P09017.

[9] Wang, K, et al. "Slip boundary condition for lattice Boltzmann modeling of liquid flows." Computers & Fluids 161 (2018): 60-73.

[10] Chen, Li, Yang Yu, and Guoxiang Hou. "Sharp-interface immersed boundary lattice Boltzmann method with reduced spurious-pressure oscillations for moving boundaries." Physical Review E 87.5 (2013): 053306.