Lattice Boltzmann Simulation of Flow Past a Finite Cylinder

Yi Liu, Deming Nie
Institute of Fluid Mechanics, China Jiliang University, China
nieinzh@cjlu.edu.cn

Abstract. In this work the lattice Boltzmann method was adopted to simulate the flow past a finite cylinder which is mounted on a wall. The Reynolds number based on the height of the cylinder is ranging from 10 to 160. The effect of the Reynolds number on the flow pattern as well as the drag and lift coefficient of the cylinder was discussed. Results show that the flow becomes unsteady when the Reynolds number is larger than 60. Moreover, the flow pattern becomes more complex for larger Reynolds number.

1. Introduction
Flow past a cylinder is a topic of great academic and practical importance [1]. It is well known that Karman vortex street generally takes place when flow past a fixed circular cylinder, which leads to the oscillation of the drag and lift force experienced by the cylinder. Therefore, the investigation of the flow past a cylinder helps to understand the fundamental physical phenomena of bluff body aerodynamics. Things may become much more complex when the cylinder is mounted on a wall, since the effect of the wall is significant in most cases. The flow around a cylinder of finite length is representative of flows in various applications, such as high-rise buildings, chimneys and tube banks in heat exchangers.

Due to its importance a few experimental and numerical work have been devoted to investigate the flow past a finite cylinder which is mounted on a wall. Liu et al. [2] numerically investigated the flow past a finite circular cylinder. They revealed that the tip vortex is generated at the free-end while the horse-shoe vortex is generated at the plate-body junction for the Reynolds number ranging from 100 to 200. Wang and Zhou [3] studied the wake of flow around a finite-length square cylinder at Reynolds number of 221 and 9300 through an experimental work. They observed two types of arch type vortices, which are depending on the aspect ratio of cylinder. Bourgeois et al. [4] conducted an experimental study on the turbulent flow past a finite square cylinder which is mounted on a thin boundary layer. They presented the coherent and incoherent flow fields and investigated the effect of incoherent turbulence on the turbulent characteristics. Recently, Saha [5] performed direct numerical simulation to study flow past a finite square cylinder at a Reynolds number of 250. It was found that the effect of the cylinder aspect ratio on the flow characteristics is significant. Moreover, his work revealed that the Strouhal number and drag coefficient increases as the cylinder aspect ratio increases.

So far the largest and maybe the most realistic suspension simulations have been performed with the lattice Boltzmann method (LBM) [6, 7] that appears to be advocated as an effective computational tool for the simulation of particle suspensions, multiphase flow, microfluidics, and turbulence for its several remarkable advantages, such as easy coding, no requirement of re-meshing procedure, and high computational efficiency.

In comparison with the research on the flow past infinite cylinder, the number of research on finite length cylinder is limited. As mentioned, there are only a few numerical simulations of flows around
tall cylinders and there are still open questions about parts of this flow. For example, the flow pattern before the cylinder is not clear. In addition, the lower part of the near wake is not fully understood. The main objective of the present study is to numerically investigate the flow past a finite cylinder which is placed on a wall in a direct numerical simulation framework for the Reynolds number 10 ≤ Re ≤ 160. This range of Re is consistent with most of previous research work on flow past an infinite cylinder. This work mainly focuses on the effects of the Reynolds number on the flow pattern and drag and lift coefficient of the cylinder.

2. Numerical Model

We solve the motion of a fluid using the LBM. The discrete lattice Boltzmann equations of a single-relaxation-time model are expressed as [6]

\[ f_i(x + e_i \Delta t, t + \Delta t) - f_i(x, t) = -\frac{1}{\tau} \left[ f_i(x, t) - f_i^{(0)}(x, t) \right] \]  

where \( f_i(x, t) \) is the distribution function for the microscopic velocity \( e_i \) in the ith direction, \( f_i^{(0)}(x, t) \) is the equilibrium distribution function, \( \Delta t \) is the time step of the simulation, \( \tau \) is the relaxation time, \( c_s \) is the speed of sound, and \( w_i \) are weights related to the lattice model.

The fluid density \( \rho \) and velocity \( u \) are determined by the distribution function

\[ \rho = \sum_i f_i, \quad \rho u = \sum_i f_i e_i \]  

For the two-dimensional D2Q9 lattice model used here, the discrete velocity vectors are

\[ e_i = \begin{cases} (0,0), & \text{for } i = 0, \\ (\pm 1,0) \mathbf{c}, & \text{for } i = 1 \text{ to } 4, \\ (0, \pm 1) \mathbf{c}, & \text{for } i = 5 \text{ to } 8, \end{cases} \]  

where \( \mathbf{c} = \Delta x / \Delta t \), and \( \Delta x \) is the lattice spacing. The speed of sound \( c_s^2 = c^2 / 3 \). Following Qian et al. [11], the equilibrium distribution function is chosen as

\[ f_i^{(0)}(x, t) = w_i \rho \left[ 1 + \frac{3 e_i \cdot u}{c^2} + \frac{9 (e_i \cdot u)^2}{2c^4} - \frac{3 u^2}{2c^2} \right] \]  

where the \( w_i \) are set to \( w_0 = 4/9 \), \( w_{1,4} = 1/9 \), and \( w_{5,8} = 1/36 \). In this model, the fluid viscosity is computed using the equation \( \nu = c_s^2 (\tau - 0.5) \Delta t \).

![Figure 1. Two-dimensional physical model.](image)

The physical model considered for the analysis is schematically shown in Fig. 1. \( w \) is the width of the cylinder and \( h \) is the height of the cylinder. In the simulations, we set \( w = 20 \) (in lattice unit) and \( h = 60 \) (in lattice unit). The computational domain is \( W \times H = 30h \times 20h \) in this work. The cylinder is 10h away from the inlet. The boundary conditions employed for the present investigation are shown in Fig. 1. The Reynolds number is defined as \( Re = U_0 h / \nu \), where \( \nu \) is the kinematic viscosity of the fluid.

3. Results

Fig. 2 shows the computational results in the vicinity of the cylinder by the streamlines plots at \( Re = 10, 20, 40 \) and 60. All the cases shown in Fig. 2 are steady state. Fig. 2 shows that the flow separates at the corner of the cylinder for all Reynolds numbers. As a result, a closed steady recirculating region
consisting of one vortex forms behind the cylinder. This recirculating region increases in size with the increase in Reynolds number.

![Figure 2. Streamline for different Reynolds number: (a) Re = 10, (b) Re = 20, (c) Re = 40 and (d) Re = 60.](image)

The instantaneous streamlines in Fig. 3 show the detailed views of the laminar vortex near the cylinder at different Reynolds number. In addition to the primary separation region, there is a secondary separation region behind the cylinder, which is much smaller than the primary one. Furthermore, another separation region is observed before the cylinder for all cases, which is an unusual phenomenon. It is also clear that the size of the separation region is not sensitive to the Reynolds number.

![Figure 3. Recirculation regions for different Reynolds number: (a) Re = 10, (b) Re = 20, (c) Re = 40 and (d) Re = 60.](image)

The cylinder is experiencing the drag force $F_D$, lift force $F_L$ and torque $F_M$ resulted from the flow past on it, which are generally depending on the Reynolds number. For the sake of comparison, all the results are normalized, i.e. $C_D = F_D / 0.5 \rho U_0^2 h$, $C_L = F_L / 0.5 \rho U_0^2 h$ and $C_M = F_M / 0.5 \rho U_0^2 h^2$. As shown in Fig. 4, all results eventually reach a steady state after a damped oscillation of initial transients for all cases. The drag coefficient $C_D$ decreases as $Re$ increases. The same is true for the torque coefficient $C_M$, as shown in Fig. 4. However, the lift coefficient $C_L$ increases as $Re$ increases. In addition, the lift coefficient is much larger than the drag coefficient.

![Figure 4. Normalized force experienced by the cylinder for different Reynolds numbers: (a) drag, (b) lift and (c) torque.](image)

The instantaneous streamlines in Fig. 5 show the detailed views of the laminar vortex shedding near the cylinder at $Re = 100$, for four successive moments of time which span over the whole period.

![Figure 5. Streamline at different times for $Re = 100$: (a) $t = 10000$, (b) $t = 14000$, (c) $t = 18000$ and (d) $t = 22000$.](image)
The results shown in Fig. 5 indicate that the flow does not reach a steady state. The vortex shedding is observed at this Reynolds number. In addition, two successive vortices will come to contact and merge into a larger vortex, as shown in Fig. 5. Fig. 6 shows the instantaneous vortex contour at different times at \( Re = 100 \), which clearly display the entire evolution of vortex behind the cylinder. Each time the vortex shedding takes place, the vortex will move to the downstream, as shown in Fig. 6.

![Figure 6](image6.png)

**Figure 6.** Instantaneous vortex contour at different times for \( Re=100 \): (a) \( t = 10000 \), (b) \( t = 14000 \), (c) \( t = 18000 \) and (d) \( t = 22000 \).

Things may become more complex if increasing the Reynolds number. As shown in Fig. 7, the instantaneous streamline patterns of \( Re = 120 \) are different from those of \( Re = 100 \) shown in Fig. 5. A small vortex attached on the bottom wall is observed in the figure, which is located between two primary shedding vortices. Another significant difference is that the period is about 12000 time step for \( Re = 100 \), while it is about 28000 time step for \( Re=120 \). Moreover, the unsteadiness of flow can be demonstrated by showing the recirculation regions near the cylinder, as one can see in Fig. 8. During a single period, the size of the recirculation region before the cylinder decreases firstly, then increases. This process is repeated. In addition, this is also true for the secondary recirculation region behind the cylinder.

![Figure 7](image7.png)

**Figure 7.** Streamline at different times for \( Re = 120 \): (a) \( t = 10000 \), (b) \( t = 18000 \), (c) \( t = 28000 \) and (d) \( t = 38000 \).

![Figure 8](image8.png)

**Figure 8.** Recirculation regions at different times for \( Re = 120 \): (a) \( t = 10000 \), (b) \( t = 18000 \), (c) \( t = 28000 \) and (d) \( t = 38000 \).

The flow pattern becomes much more complex when the Reynolds number is \( Re = 160 \), which is the largest Reynolds number studied in this work. The streamlines are shown in Fig. 9 and Fig. 10. In comparison with the results of \( Re = 120 \), more recirculation regions are generated behind the cylinder. Furthermore, the recirculation regions before and behind the cylinder are much larger, as shown in Fig. 10, which may lead to higher pressure drop and larger oscillations in drag and lift force experienced by the cylinder.

![Figure 9](image9.png)

**Figure 9.** Streamline at different times for \( Re = 160 \): (a) \( t = 10000 \), (b) \( t = 18000 \), (c) \( t = 28000 \) and (d) \( t = 38000 \).
Fig. 11 shows the time history of normalized drag force, lift force and torque experienced by the cylinder for different Reynolds numbers. Different from the results of small Reynolds numbers shown in Fig. 4, all the results are oscillating even after a long time, which is resulted from the vortex shedding behind the cylinder. For all cases the magnitude of oscillation increases as the Reynolds number increases. Similarly, the lift coefficient is much larger than the drag coefficient. And the normalized torque is larger than $C_D$ and $C_M$ for several orders of magnitude.

![Figure 10](image1)
![Figure 11](image2)

**Figure 10.** Recirculation regions at different times for $Re = 160$: (a) $t = 10000$, (b) $t = 18000$, (c) $t = 28000$ and (d) $t = 38000$.

**Figure 11.** Normalized force experienced by the cylinder for different Reynolds numbers: (a) drag, (b) lift and (c) torque. Red: $Re = 100$, green: $Re = 120$, purple: $Re = 140$ and blue: $Re = 160$.

### 4. Conclusion

In this work, the lattice Boltzmann method was adopted to simulate the flow past a finite cylinder for the Reynolds number $10 \leq Re \leq 160$. The streamline pattern as well as the drag and lift force was presented. For small Reynolds number, the flow can reach a steady state eventually. The recirculation regions near the cylinder are displayed. The drag coefficient $C_D$ decreases as $Re$ increases. The same is true for the torque coefficient $C_M$. However, when the Reynolds number is increasing, the flow becomes unsteady, which leads to the vortex shedding behind the cylinder. Especially, there is a small recirculation region which is attached on the bottom wall when $Re$ is large enough. In addition, results also show that the drag and lift coefficients are oscillating with large magnitude.

### Acknowledgments

This work was supported by the Zhejiang Provincial Natural Science Foundation of China (Grant LY15A020004)

### References

[1] S. Kang, Phys. Fluids 15 (2003).
[2] Y. Liu Y, R.MC. So, Z.X. Cui, J. Fluids Struct. 20 (2005).
[3] H.F. Wang, Y. Zhou, J. Fluid Mech. 638 (2009).
[4] J.A. Bourgeois, P. Sattari, R.J. Martinuzzi, Phys. Fluids 23 (2011).
[5] A. K. Saha, Comput. Fluids 88 (2013).
[6] S.Y. Chen, G.D. Doolen, Annu. Rev. Fluid Mech. 30 (1998).
[7] C.K. Aidun, J.R. Clausen, Annu. Rev. Fluid Mech. 42 (2010).