Numerical Simulation of Flow Around a Row of Circular Cylinders Using the Lattice Boltzmann Method

S. Ul Islam and C.Y. Zhou
Harbin Institute of Technology, Shenzhen Graduate School, Shenzhen University Town, Shenzhen 518055, China

Abstract: This study describes a numerical study of flow past a row of circular cylinders at different Reynolds numbers with different distances between the cylinders using the Lattice Boltzmann Method (LBM). Numerical simulations are performed to investigate the blockage effect for the ranges of $R_e\leq200$ and $B=W[R_e]\leq25R_e$, where, $R_e$, $R$, and $W$ are the Reynolds numbers, the radius of the cylinders and the distance between the center of the cylinders, respectively. The Strouhal number and drag forces exerted on the cylinders are quantified jointly with the flow patterns around the cylinders in the form of vorticity contours. It is found that both the drag coefficient and Strouhal number increase when $B$ decreases. It is also observed that the Strouhal number, in general increases as $R_e$ increases for a fixed value of $B$ for the ranges of $R_e$ and $B$ studied. The distance $B$, between cylinders is limited within 25 in this simulation because of computational resources.

Key words: Lattice Boltzmann method, blockage effect, drag coefficient, strouhal numbers, row of circular cylinders

INTRODUCTION

The flow past a row of circular cylinders is an area of considerable interest to numerical analyst as well as theoreticians. A vast body of knowledge encompassing analytical, numerical and experimental studies is now available on the flow past a circular cylinder (Zdravkovich, 2003). This study will shed light between the drag, blockage ratio (distances between centers of cylinders), Reynolds number and the Strouhal numbers for the two-dimensional flow past a row of circular cylinder. Systematically studied for large distances between centers of the cylinders was conducted by Fornberg (1991).

Over the past few decades, much theoretical and numerical effort has been made to investigate the distances between centers of cylinders (Boppana and Gajjar, 2007; Gajjar and Nabila, 2004; Fornberg, 1998; Natarajan et al., 1993; Smith, 1992). Gajjar investigated instability of flow past a cascade of circular cylinders (Gajjar, 2006).

The interface effect between the cylinders in the row is a very remarkable at low Reynolds numbers (Tamada and Fujikawa, 1957). A steady flow through a uniform cascade of normal flat plates numerically and experimentally studies (Ingham et al., 1990). However, the studies, especially on numerical simulations of flow past a row of circular cylinders are still relatively partial. Therefore, the further effort is made in this study to study numerically the flow past a row of circular cylinders. In the present study, the Lattice Boltzmann Method (LBM) is employed to simulate such flows past a row of circular cylinders. LBM is originated from the Lattice Gas Cellular Automata (LGCA) (Frisch et al., 1986, 1987). In LGCA, the fictitious particles move along lattices with streaming and collision. However, LGCA suffers some drawbacks such as large statistical noise, unphysical velocity-dependent pressure, non-Galilean invariance and large numerical viscosities. It is the lattice Boltzmann method that overcomes the drawbacks of LGCA. In LBM, the Bhatnagar-Gross-Krook (BGK) (Bhatnagar et al., 1954) collision model used in the standard Boltzmann equation is adopted.

In recent years, the Lattice Boltzmann Method (LBM) has received considerable attention as an alternative numerical scheme for simulating a variety of fluid dynamic problems (Chopard and Droz, 1998; Wolf-Gladrow, 2000; Succi, 2001; Benzi et al., 1992; Chen and Doolen, 1998). A two-dimensional nine-velocity (D2Q9) LBM model (Xiaoyi and Li-Shi, 1997) has been used in this study.

The purpose of the present study is an attempt to examine the effect of distances between centres of cylinders up to Reynolds number of 200. A little information is available about the effect of distances...
between centers of cylinders on drag and wake characteristics. On the other hand, much information is available on the drag, transition from one regime to another, vortex shedding, etc. The numerical study has been carried out using the Lattice Boltzmann Method (LBM). This study shows the capability of the proposed model and the interface effect between the cylinders in the row is very remarkable at low Reynolds numbers.

**LATTICE BOLTZMANN METHOD**

A square lattice D2Q9 with unit spacing is used in which each node has eight nearest neighbors connected by eight links (Fig. 1). Since, the details of D2Q9 model are given in detail there (Xiaoyi and Li-Shi, 1997) for incompressible Navier-Stokes (NS) equations independent of the density variation. Some basic features of the proposed LBM alone are presented here. Hence, there are two types of moving particles. Particles move along the axis with speed $e_i = 1, i = 1,2,3,4$, particles move along the diagonal directions with speed $e_i = \sqrt{2}, i = 5,6,7,8$

Rest particles also allowed at each node with speed zero. The velocity directions of these particles are defined as:

$$e_i = \begin{cases} (0,0), & i = 0 \\ \left(\cos\left(\frac{\theta - 1}{2}\pi\right), \sin\left(\frac{\theta - 1}{2}\pi\right)\right), & i = 1,2,3,4 \\ \left(\frac{\cos\left(\frac{\theta - 1}{4}\pi\right)}{2}, \frac{\sin\left(\frac{\theta - 1}{4}\pi\right)}{2}\right), & i = 5,6,7,8 \end{cases}$$  

where, $c = \frac{\delta x}{\delta t}$, $\delta x$ and $\delta t$ are the lattice constant and the time step size, respectively.

The particle distribution function satisfies the following evaluation equation of the system is:

$$P_i(x,t+\delta t) = P_i(x,t) + \Omega_i$$  

where, $\Omega_i$ is the collision operator representing the rate of change of the particle distribution function due to collision and $P_i(x,t)$ is the density distribution function. The lattice Boltzmann BGK equation (in lattice units) after Bhattacharji-Gross-Krook (BGK) (Bhatnagar et al., 1954), with single-time relaxation approximation is:

$$P_i(x,t+\delta t) = P_i(x,t) - \frac{1}{\tau_i} [P_i(x,t) - P_i^{eq}(x,t)]$$  

where, $P_i^{eq}$ is the equilibrium pressure distribution at $x$, $t$ and $\tau_i$ is the relaxation time which controls the rate of approach to equilibrium. The pressure $p$ and the macroscopic velocity $u$ are defined in terms of the particle distribution function by:

$$p = \sum_i p_i$$  

and

$$p_i u = \sum_i p_i u_i$$  

The corresponding equilibrium pressure distribution $P_i^{eq}$ defined by:

$$P_i^{eq} = w_i \left[ p_i^{eq} = \frac{1}{\tau_i} \left( \frac{3}{2} u_i^2 - \frac{1}{2} \frac{3 w_i u_i}{c^2} \right) \right]$$  

where, $p = \frac{\rho}{\rho_i}$ and $\rho_i$ is the constant average density. The weighting factor $w_i$ is given by:

$$w_i = \begin{cases} \frac{4}{9}, & i = 0 \\ \frac{1}{9}, & i = 1,2,3,4 \\ \frac{1}{36}, & i = 5,6,7,8 \end{cases}$$  

The speed of sound in this model is:

$$c_s = \frac{c}{\sqrt{3}}$$
In order to derive the Navier-Stokes (NS) equations from LBE, under the incompressible flow limit i.e., the Mach number

\[ M_i = \frac{|u|}{c_s} \ll 1 \]

the Chapman-Enskog expansion (Chapman et al., 1999) is used, in essence with a standard multi-scale expansion proposed by Frisch et al. (1987), the mass and momentum equations can be derived from the D2Q9 model as follows:

\[ \nabla \cdot u = 0 \]  
\[ \frac{\partial u}{\partial t} + (u \cdot \nabla)u = -\nabla p + \nu \nabla^2 u \]  

where, the kinematic viscosity, \( \nu \), is:

\[ \nu = \frac{(2\tau - 1) \delta^2}{6 \Delta t} \]

Since, the details of this model are given by Xiao and Li-Shi (1997).

**PROBLEM STATEMENT AND FORMULATION**

The computational domain is drawn in Fig. 2. A two-dimensional flow past a row of circular cylinders with radius \( R \) is investigated, where five different values of \( B= W/R \), i.e., 8R, 10R, 15R, 20R and 25R are examined. A rectangular computational domain of \( B \times 68R \) (width \times length) is selected with one cylinder lying on the horizontal central line and symmetric boundary condition is applied at the top and bottom boundaries of the computational domain to represent a row of cylinders. Five different lattices \( 60 \times 504, 74 \times 504, 112 \times 504, 148 \times 504 \) and \( 168 \times 504 \) are used for the five values of \( B \), respectively. The relaxation parameter \( \tau \) is set to be 0.5173, 0.5086 and 0.5043, respectively.

**BOUNDARY CONDITIONS**

**Initial condition:** Potential flow solution is adopted as an initial condition.

**Inlet boundary:** A constant pressure gradient along the x-direction is imposed (Luo, 1998):

\[ f_i(x, t) = f_i(x, t) - \frac{3\nu}{c_s^2} \frac{\partial}{\partial x} \tilde{s}_i \tilde{\xi} \]

(11)

where, \( f_i \) and \( \tilde{s}_i \) are post-collision of distribution function and unit vector along the x axis, respectively.

**Outlet boundary:** A simple extrapolation is adopted for exit.

\[ f_i(N_x \pm 1,i) - f_i(N_x \pm 2,i) \]

(12)

where, \( N_x \) is the number of lattices in the x-direction.

**Surface of the cylinder:** No-slip wall boundary condition is applied to the surface:

\[ u = 0; v = 0 \]

(13)

and this is realized with a bounce-back boundary treatment.

**Top and bottom boundaries:** On the top and bottom boundaries symmetry boundary condition is applied by Renwei (1999).

\[ f_i(i,1) = f_i(i,3), \quad f_i(i,1) = f_i(i,3), \quad f_i(i,1) = f_i(i,3) \]

(14)

\[ f_i(i,1) = f_i(i,3), \quad f_i(i,1) = f_i(i,3), \quad f_i(i,1) = f_i(i,3) \]

\[ f_i(i,1) = f_i(i,3), f_i(i,1) = f_i(i,3), f_i(i,1) = f_i(i,3) \]

**Force evaluation:** The momentum exchange method proposed by Dazhi et al. (2003) was used in the present study for simulating the flow around the circular cylinder. Since, the details of this model are given here.

\[ F = \sum_{a \neq b} \sum \delta \frac{1}{2} \left[ f_i(x, i) + f_i(x + e_b, i, t) \right] \left[ 1 - w(x + e_b) \right] \frac{\partial}{\partial t} \]

(15)

Reynolds number \( R_e \) is defined by:

\[ R_e = \frac{U_{\infty}D}{\nu} \]

(16)

where, \( D \) is the cylinder diameter. Other important dimensionless numbers are the drag coefficient \( C_D \), the lift coefficient \( C_L \), the Strouhal number \( S_t \) and blockage ratio \( B \). They are defined by the following formulas:

---

Fig. 2: Schematic flow configuration
where, $f_v$ is the vortex shedding frequency from the cylinder, $F_x$ and $F_y$ are the force components in the in-line and transverse directions, respectively.

Computation is terminated when the following convergence criteria is satisfied:

$$\frac{\sum |u_n^{(m)} - u_n^{(m-1)}|}{\sqrt{\sum |u_n^{(m-1)}|^2}} \leq 1 \times 10^{-7}$$  \hspace{1cm} (21)

The proposed LBM model (Xiaoyi and Li-Shi, 1997) is independent of density variation. Instead of Eq. 17 and 18 we used Eq. 22 and 23 for calculating drag and lift coefficients (because the density, $\rho$, is here the average density and is set to be 1.0):

$$C_d = \frac{F_d}{1/2 \rho U^2 \cdot D}$$ \hspace{1cm} (22)

$$C_l = \frac{F_l}{1/2 \rho U^2 \cdot D}$$ \hspace{1cm} (23)

**RESULTS**

The graphical representation of different compositions and comparison with literature data (Fornberg, 1991) shown here. Figure 3 shows how the drag coefficients vary with different blockage ratios.

Figure 4 gives similar behavior for drag coefficient with different Reynolds number ($R_e$).

The structures of the vorticity filed are shown in Fig. 5a and b.

Figure 6a and b show the vortex formation for different blockage ratios.

The spectrum analysis for the lift coefficient displayed graphically in Fig. 7a-f.

Figure 8-11 show the variation of Strouhal numbers for different blockage ratios, Strouhal number variation for different Reynolds number $R_e$ and similarly examine the effect of drag coefficients for different Reynolds number $R_e$ and blockage ratios.
Fig. 6: Vorticity contours of vortex wake of the cylinder for different blockage ratio with (a) $R_e = 100$ and (b) 200

Fig. 7: Continued
Fig. 7: Spectrum analysis of lift force time history, (a) $B = 10$, $R_c = 50$, (b) $B = 10$, $R_c = 100$, (c) $B = 10$, $R_c = 200$, (d) $B = 25$, $R_c = 50$, (e) $B = 25$, $R_c = 100$ and (f) $B = 25$, $R_c = 200$

Fig. 8: Variation of $S_i$ with $B$ for different $R_c$

Fig. 9: Variation of $S_i$ with $R_c$ for different $B$

Fig. 10: Variation of $C_s$ with $R_c$ for different $B$
The results of the spectrum analysis are then shown in Fig. 8 and 9. It can be seen that the vortex shedding frequency decreases as the blockage ratios increases. In other words, the vortices shed with a higher frequency when the distance between the cylinders becomes smaller. The shedding frequency goes up as Reynolds number increases, which shows the general behavior of $S_t$ with $R_e$ for this Reynolds number range. This is similar to an isolated cylinder in a uniform flow as expected. Figure 10 and 11 show that the drag coefficient goes up as Reynolds number decreases and drag coefficient decreases when the blockage ratios increases with fixed Reynolds number.

In practice, the flows past a row of circular cylinders are usually more important and are also hard to incorporate. Therefore, we put more emphasis on them in this study to numerically investigate such types of flows. There is a slight difference with the present and literature data (Fornberg, 1991). Many possibilities are here: different numerical method, limitations of the present method, inflow boundary conditions, side wall boundary conditions and so on.

**CONCLUSION**

A numerical investigation on a uniform flow past a row of circular cylinders is conducted using the lattice Boltzmann method. The effects of the blockage ratio on the drag coefficient, the vortex wake and vortex shedding frequency are examined. The results show that both the drag coefficient and the vortex shedding frequency increase as the cylinders becomes closer. The present results show good agreement with some data in literature (Fornberg, 1991) and indicate the capability of the LBM in simulating the flow field with complex geometries. This study also sheds the light on the effect of the numerical blockage effect on a numerical solution for an unbounded circular cylinder. When the grids with high blockage are used for reducing the computational time required, the values of the various hydrodynamic parameters may be considerably different from what they should be. When $R_e = 50$, as $B$ increases, the wake flow becomes steady. However, as $B$ decreases the alternate vortex shedding from the cylinder occurs. The current simulation shows that the proposed method is capable of handling such types of flows problems.

**ACKNOWLEDGMENT**

The study described in this research was supported by a grant from National Natural Science Foundation of China (Project No. 90715031). The financial support is gratefully acknowledged.
REFERENCES


