한국마린엔지니어링학회지 제33권 제1호, pp. 62~70, 2009. 1 (ISSN 1226-9549)
Journal of the Korean Society of Marine Engineering

Lattice-Boltzmann Simulation of Fluid Flow around a Pair of Rectangular Cylinders

M.A.Taher* · Tae-Sil Baek** and Yeon-Won Lee†

(Manuscript : Received October 23, 2008 ; Revised : November 21, 2008 ; Accepted : January 21, 2009)

Abstract : In this paper, the fluid flow behavior past a pair of rectangular cylinders placed in a two dimensional horizontal channel has been investigated using Lattice-Boltzmann Method (LBM). The LBM has built up on the D2Q9 model and the single relaxation time method called the Lattice-BGK (Bhatnagar-Gross-Krook)model. Streamlines, velocity, vorticity and pressure contours are provided to analyze the important characteristics of the flow field for a wide range of non dimensional parameters that present in our simulation. Special attention is paid to the effect of spacing (d) between two cylinders and the blockage ratio $A(=h/H)$, where H is the channel height and h is the rectangular cylinder height, for different Reynolds numbers. The first cylinder is called upstream cylinder and the second one as downstream cylinder. The downstream fluid flow fields have been more influenced by its blockage ratios (A) and Reynolds numbers (Re) whereas the upstream flow patterns(in front of downstream cylinder) by the gap length (d) between two cylinders. Moreover, it is observed that after a certain gap, both upstream and downstream flow patterns are almost similar size and shape. The simulation result has been compared with analytical solution and it is found to be in excellent agreement.

Key words : Lattice-Boltzmann, Bhatnagar-Gross-Krook, Reynolds number, Rectangular cylinder

1. Introduction

Lattice-Boltzmann Method (LBM) has evolved as an alternative numerical approach for the solution of a large class of problems over the last decade. In Computational Fluid Dynamics (CFD), fluid properties, such as density, pressure, velocity and temperature are typically described by the N-S equations, which have nonlinear terms making them too expensive to solve numerically in real time. Unlike N-S solvers, LBM does not need to solve partial differential equations. It only involves algebraic operation and easy to implementation.

† Corresponding Author(School of Mechanical Engineering, Pukyong National University, E-mail: ywlee@pknu.ac.kr, Tel: 051)629-6162)
* Graduate School of Mechanical Engineering, Pukyong National University
** Department of Automobile, Pohang college

However, the LBM has demonstrated a significant potential and broad applicability with numerous computational advantages to incorporate microscopic interactions. In this method, the grid involves two types of rules: propagation and collision. Propagation means the microscopic particles move to the nearest neighbor along their velocity direction. Collision is the most important part: it can force particles to change direction and is decided by the collision operator. Particularly, a simple linearized version of the collision operator makes use of a relaxation towards an equilibrium value using a single relaxation time parameter known as BGK model. Lattice gas models with an appropriate choice of the lattice symmetry in fact represent numerical solutions of the Navier-Stokes equations and therefore able to describe the hydrodynamics problems discussed by McNamara and Zanetti[1] and Wei et al.[2]. It is recognized that the LBM can be used to simulate the incompressible N-S equations with high accuracy and this lattice BGK model, the local equilibrium distribution has been chosen to recover the N-S macroscopic equations by many authors among them Chen et al.[3], He and Luo[4], and Qian et al.[5]. An overview of LBM, a parallel and efficient algorithm for simulating single-phase and multiphase fluid flows and also for incorporating additional physical complexities have been discussed by Chen and Doolen[6]. There is no doubt that LBM has several advantages over other conventional CFD methods, especially in dealing with complex boundaries, incorporating of microscopic interactions, are described in the excellent books by Succi[7] and Sukop[8]. It is no exaggeration to say that an enormous (and still rapidly growing) corpus of literature on the subject of bluff body wake has been developed since the pioneering work of Von K'arman early last century. Actually the flow past different bluff bodies is of immense very important in the design of offshore structures, marine vessels, pipelines and water based generation system, e. g. vortex flowmeters, building, bridges, towers etc. The simulation of flow behavior around bluff bodies both in numerical and experimental using different methods have been studied very recently by Cheng et al.[9], Inoue et al.[10],J.W Kimet al.[11] and E.R.Kimet al.[12]. The objective of this paper is to numerically study the fluid flow behavior around a pair of rectangular cylinders using LBM, where the flow can be driven with the pressure (density) gradients. Here we have focused our attention on the evolution of streamlines, vorticity contours, pressure contours as well as velocity profiles to investigate the important characteristics of the flow field around bluff bodies for a wide range of non-dimensional parameters. Throughout our calculation we use Lattice-Boltzmann units. In this simulation Δx and Δt are assumed as the space(lu) and time unit(ts) respectively.

2. Formulation of the Problem

Consider two-dimensional steady compressible flow around a pair of rectangular cylinders placed in a channel as shown in Fig.1.

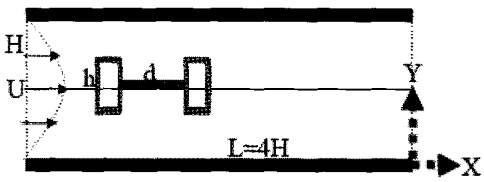


Fig. 1 Physical model and coordinate systems

The computational domain is to consider as a rectangular region $L \times H$, where H is the height and L is the length of the channel. The height of the cylinder is denoted by h and length of the cylinder is $l=h/2$. In an incompressible flow, Reynolds number is the only parameter that controls the flow field and is define by $Re=UH/v$, where U and H are the characteristic velocity and the length respectively. Computations have been carried in two different blockage ratios, A=0.20 and 0.40 with the gap between cylinders, d/h=0.0,0.10, 0.20, 0.30 and 0.40, for the Reynolds number ranging from 0 to 450 based on the characteristic length of the channel, the maximum incoming flow velocity (less then 0.1 *lu*) and also the nature of fluid transport properties. It is noted that, if $d=0$, then two rectangular cylinders becomes a square cylinder. If a bluff body is placed in a flow stream, vortices from behind shed alternately from each side. In fluid dynamics, vorticity is the circulation per unit area at a point in the flow field. Mathematically, it is defined as, $\vec{w}=\nabla \times \vec{u}$, where $\vec{u}$ is the fluid velocity.In order to simulate a fully developed laminar flow, a parabolic velocity profile can be expressed as

$$u(y) = U[1-(1-2y/H)^2] \quad (1)$$

And $v = 0$ with a maximum velocity U at the channel inlet. This velocity is chosen to be lower than 10% of the speed of sound for LBM simulation to avoid significant compressibility effects. In our simulation, it has been used Zou-He Boundary condition to implement Dirichlet boundaries on inlet/outlet by Sukop and Thorne [8]. In the top and bottom wall, no slip boundary condition imposed by the standard full bounce-back treatment. The macroscopic variables of interest (i.e. density, pressure, velocity) can easily be obtained from the particle distribution function f_i. Neglecting external forces ,the transport equation for f_i can be expressed by the Boltzmann equation (BE) with BGK approximation as

$$\frac{\partial f_i}{\partial t}+\vec{e}_i \cdot \frac{\partial f_i}{\partial \vec{x}_i}=-\frac{1}{\tau}(f_i-f_i^{eq}),\ i=0,1,\text{---}q-1 \quad (2)$$

Where $\vec{f_i}(\vec{x},t)$ is the velocity discrete particle distribution function, f_i^{eq} is the discrete equilibrium distribution function, $\vec{x}$ is the spatial position vector, τ is the relaxation parameter and $\vec{e_i}$ is the finite set of particle velocity vectors. The total number of discrete velocities on each node in D2Q9 model is 9. The velocity of the particles are such that they move from one node to another node during each time step. These particles velocity vectors are shown in the following table:

Table 1 D2Q9 lattice velocities

e_0 =(0,0)	e_1=(c,0)	e_2=(0,c)	e_4=(0,-c)	e_5=(c,c)
e_6=(-c, c)	e_7=(-c,-c)	e_8=(-c,c)		

Here $c = \Delta x/\Delta t$ is called CFL umber. The discrete form of equation (2) is called the Lattice Boltzmann equation (LBE) and can be written as

$$f_i(\vec{x}+\Delta t\vec{e_i},t+\Delta t)-f_i(\vec{x},t)=-\frac{1}{\tau}(f_i-f_i^{eq}) \quad (3)$$

The general form of the equilibrium distribution function is defined by

$$f_i^{eq}=\rho w_i[1+\frac{3}{c^2}\vec{e_i}.\vec{u}+\frac{9}{2c^4}(\vec{e_i}.\vec{u})^2-\frac{3}{c^2}u^2] \quad (4)$$

where w_0 = 4/9, w_i = 1/9, i = 1,2,3,4 and w_i = 1/36, i = 5,6,7,8. Using the Chapman-Enskog expansion, it is mathematically provable that the equation (4) can recover the N-S equation, if the pressure and the kinetic viscosity are defined by

$$P=\rho Cs^2 \quad \text{and} \quad \nu=(\tau-\frac{1}{2})Cs^2\Delta t \quad (5)$$

where $Cs=\sqrt{RT}$ is the speed of sound. However the relaxation parameter, $\omega = 1/\tau$, depends on the local macroscopic variables, ρ and, $\rho\vec{u}$ and should satisfy the following laws of conservation:

$$\rho=\sum_i f_i^{eq} \text{ and } \rho\vec{u}=\sum_i \vec{e_i} f^{eq_i} \quad (6)$$

The above expressions describe the relationships between the microscaled quantities define on the basis of lattice sizes and the macroscaled physical quantities of flow such as the mass density and the velocity of the fluid.

3. Results and Discussion

In this problem, equation (3) is an algebraic equation and we have solved this equation on a uniform 2D grid system along with boundary conditions and other equations described in the above. Each numerical time steps consists of three stages : (i) collision, (ii) streaming and (iii) boundary conditions followed by the LBM approaches. However, in conventional CFD methods for incompressible N-S equations, we need to solve the Poisson equation for the pressure, while in LBM, solving the equation (3) we get all informations that we interested to our study. In order to assess the accuracy of our method, we compare our results (LBM) with analytical solution for Re=100, 200 as shown in Fig.2

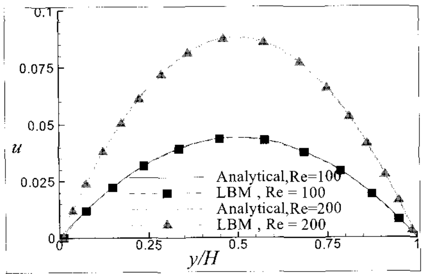
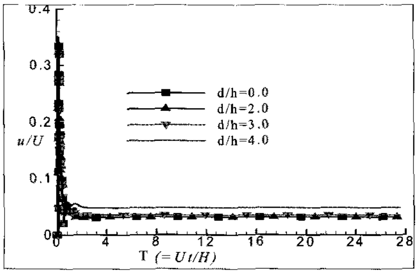


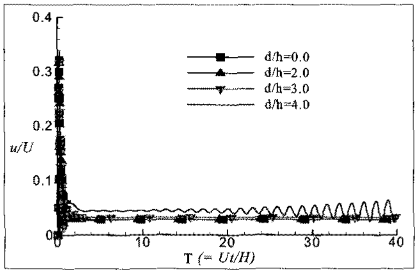
Fig. 2 Velocity in x -direction for different Re

The solid lines are represented the analytical solution and the dashes lines are the data results obtained from our simulation. It is seen that the velocity profiles in the channel are the parabolic shape and highest at the middle position of the channel. It is obviously as we have considered the fully developed laminar parabolic flow and it is seen that our results are in good agreement with analytical solution. This confirms the accuracy of our present simulation. In this model, there is a difficulty that it can take time to reach a stable state. When a flow is induced across the lattice, it will take time for the particles to distribute themselves across the lattice. This leads to fluctuations in the behavior of the flow until the model has time to settle down into a stable state.

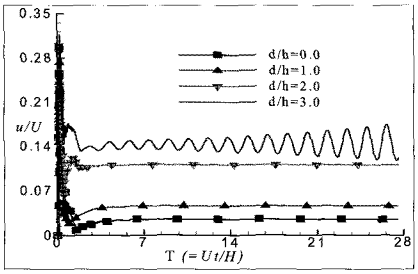
The average velocity for blockage ratios A=0.2 and 0.4 with different spicing *d/h* (=0.0, 1.0, 2.0 and 3.0) for Re= 200 and 300 respectively at the cross section of the channel are shown in Figs.3(a)-(d). Fig.3(a) shows that, for A=0.2, Re=200, the velocity took stable condition after some times and the change of fluid flow behavior is not affected by the gap length but if we increase the Reynolds number it is seen that the fluid flow is influenced by the gap length when d>3h (Fig.3(b)). However, for high blockage ratio, A=0.4,the velocity shown unstable when



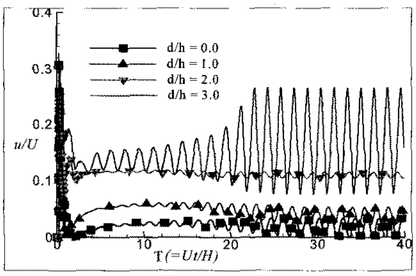
(a) A=0.2, Re=200



(b) A=0.2, Re=300



(c) A=0.4, Re=200



(d) A=0.4, Re=300

Fig. 3 Velocity plotted against non-dimensional time T(=*Ut/H*), where *t* is the lattice time steps

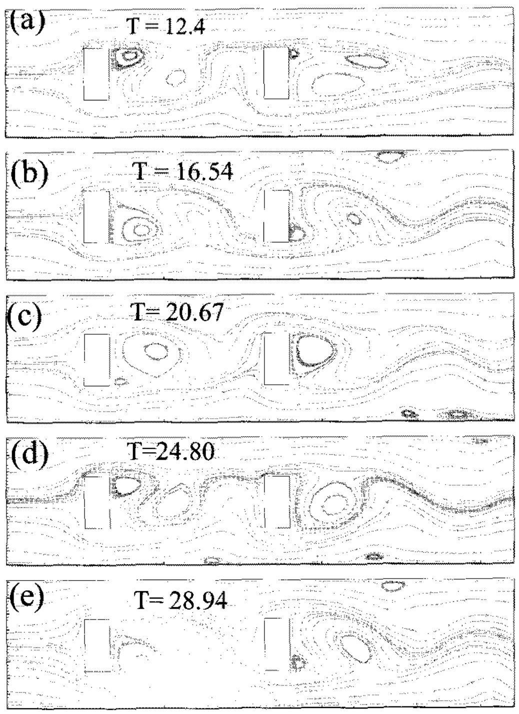


Fig. 4 Time development (T=*Ut/H*, where *t* is the lattice time steps) of vorticity for A=0.4, d/h=3.0 with Re=300

d>2h for Re=200 and if we increase the Reynolds number with high blockage ratio then for any gap length, the velocity strongly fluctuated and a simple harmonic motion occurred. To analyze the time development of vorticity with Re=300 is shown in Fig.4.

In the first plot, a top upstream vortex is forming and the low pressure core of the fully developed vortex has pulled away from the body. Three vortices are seen just behind the downstream cylinder, two is fully developed and another one is just forming. Fig.4(b) shows that the upstream top vortex is now extended itself across the entire back side of the body while the top vortex of the downstream is sliding from top to bottom cause of other vortices are pulled away from the body. From the next plots, it is observed that one vortex is forming and another one is pulled away from the body. Finally, at T=28.94,the last plot is nearly identical to the Fig. 4(b) at T=16.54. This evolutionary process is repeated approximately every T= 12.4 . A detailed view of flow field behind the cylinders and changes in the vortex shedding pattern with different blockage ratios A=0.2, 0.4 and spacing d/h=0.0,1.0,2.0,3.0 at 50,000 time steps are shown in the Figs. 5-8.

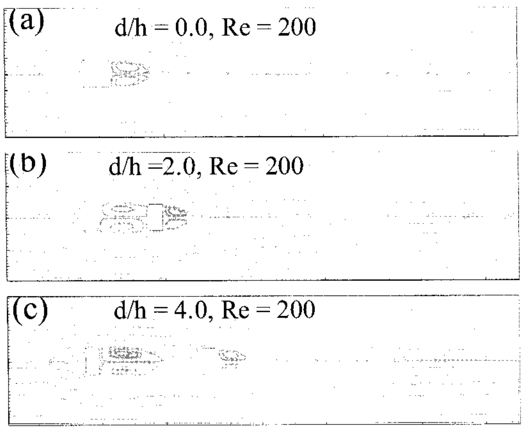


Fig. 5 Streamlines plot in x-y plane with A=0.2

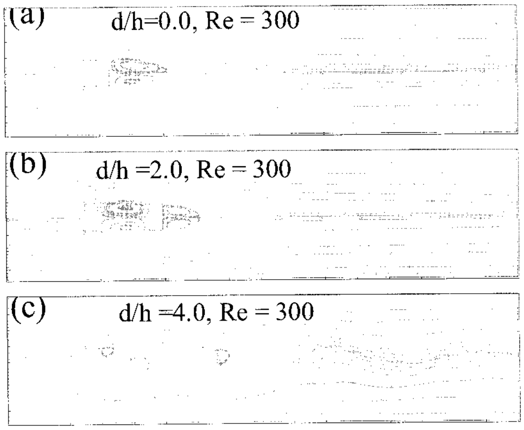


Fig. 6 Streamlines plot in x-y plane with A=0.2

From the above plots, it is seen that when d=0.0,the flow pattern is symmetric for different Reynolds numbers with respect to the on coming flow and there is a closed recirculation zone just behind the body. This zone is made up of two symmetrical vortices that rotate in opposite direction but when $d=2h$ (Fig.5(b) and 6(b)), a pair of up stream vortices have been created with the same length of the gap between two cylinders whereas the length of downstream vortices have been reduced . Further, increasing the distance between two cylinders, it is observed that for Re=200, the shape of upstream vortices only changed but for Re= 300 both up stream and downstream vortices changed abruptly. Finally both upstream and downstream vortices are almost similar size and shape for A=0.2, Re=200 when $d>4h$ and $d>3h$ for Re=300.

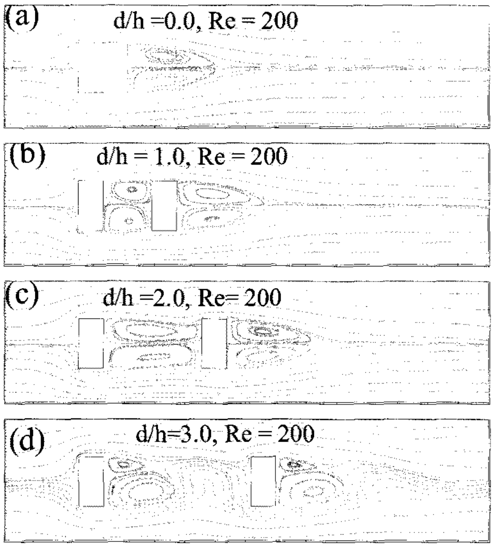


Fig. 7 Streamlines plot in x-y plane with A=0.4

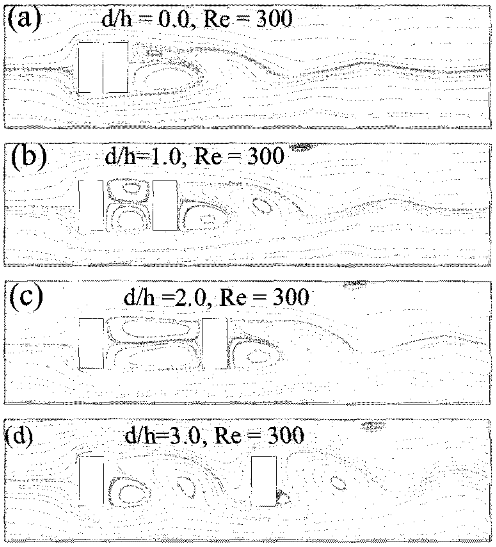


Fig. 8 Streamlines plot in x-y plane with A=0.4

The flow patterns around a pair of rectangular cylinders for A=0.4 with Re=200, 300 and different spacing d/h=0.0,0.1,0.2,0.3 are shown in Figs.7-8. It is seen that when d=0.0, a pair of vortices are form just behind the body with same strength for Re=200 (Fig.7(a)) but for higher Reynolds number (=300), the positive vortex (anticlockwise) becomes larger then negative one. It is observed that, if we increase the spacing between two cylinders, both upstream and downstream vortices changed and at a certain distance both of them are almost similar size and shape. For Re=200, it has found when $d \geq 3h$ whereas for Re=300, it has seen when $d > 2h$. Moreover, to understand the fluid flow behavior, it is important to study about the pressure distribution. Figs.9-12 show the pressure distribution for blockage ratios A=0.2 and 0.4 with different spacing and Reynolds numbers.

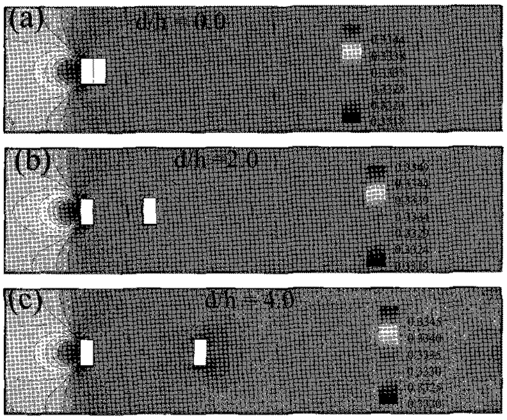


Fig. 9 Pressure field for flow with A=0.2, Re=200

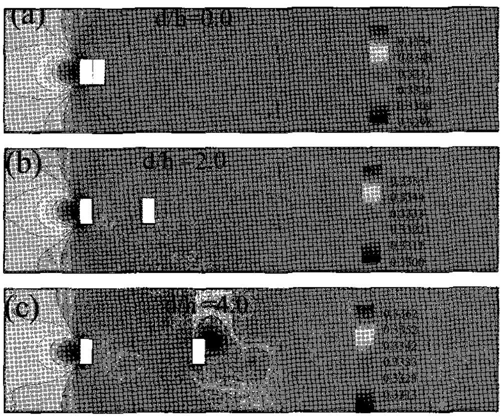


Fig. 10 Pressure field for flow with A=0.2, Re=300

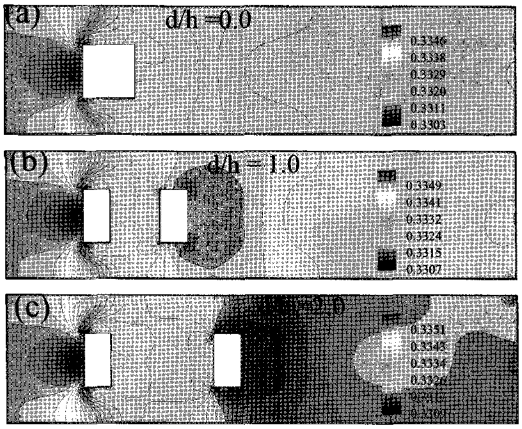


Fig. 11 Pressure field for flow with A=0.4, Re=200

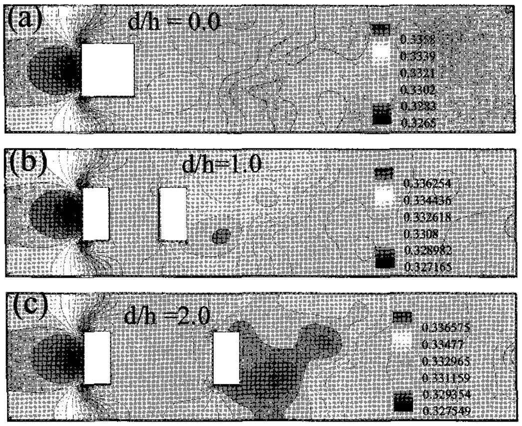


Fig. 12 Pressure field for flow with A=0.4, Re=300

It is seen from the above figures, the pressure is the highest in front of upstream cylinder and the lowest just behind the downstream cylinders. When A=0.2, the pressure drop is linear for lower Reynolds number whereas for higher Reynolds number the pressure contours show more complex pattern(Fig.10(c)) if $d \rangle 3h$. For A=0.4, the pressure drop becomes more unstable and complex (Figs.11-12) and it is observed that $d/h \geq 1.0$ when Re=200 and $d/h \geq 0.0$ when Re=300. Actually the closed pressure contours indicate the location of vortex centers, where the pressure has local minimum.

4. Conclusions

In this paper, the Lattice- Boltzmann method (LBM) has been successfully applied to simulate a two-dimensional channel flow past a pair of rectangular cylinders. The present simulation provides some important information regarding flow in the wake behind the cylinders for different Reynolds numbers, blockage ratio (A) as well as gap ratio between two cylinders. It has been observed that for low blockage ratio, the velocity has shown stable condition until a certain gap length and this gap depends on both the Reynolds number and the blockage ratio. Further, if we increase the distance, the velocity has been strongly fluctuated and a simple harmonic motion occurs. Moreover after that certain distance both upstream and downstream flow patterns are same. For high blockage ratio, wall proximity effects also observed. Consequently, the similar flow behavior observed for pressure contours. It is noted that pressure contours are more influenced by Reynolds number. The variations of the pressure contours are not significant when the recirculation flows are fully developed and it is see for low Reynolds number as well as low gap ratio between two cylinders.

References

[1] G.R. McNamara and G. Zanetti, "Use of the Boltzmann equation to simulate lattice-gas automata", Physical Review Letters, 61, pp. 2332-2335, 1988.

[2] X. Wei, K. Mueller and A. E. Kaufman, "The lattice Boltzmann method for simulating gaseous

Phenomena", IEEE Transactions on Visualization & comp. Graph,10 (2), pp. 164-176, 2004.

[3] H. Chen, S. Chen and W.H. Matthaeus, "Recovery of the Navier -Stokes equations using a lattice-gas Boltzmann method", Physical Review A, 45(8), pp. 5339-5342, 1992.

[4] X. He and L-S. Luo, "Theory of the lattice Boltzmann method: From the Boltzmann equation to the lattice equation", Physical Review E,56(8) pp. 6811-6817, 1997.

[5] Y. H. Qian, D. D'Humieres, and P. Lallemand, "Lattice BGK models for Navier-Stokes equation", Euro physics Letter,17(6),pp. 479-48, 1992.

[6] S. Chen and G. D. Doolen, "Lattice Boltzmann method for fluid flows", Annu. Rev. Fluid Mechanics, 30 pp. 329 - 364, 1998.

[7] S. Succi, "The lattice Boltzmann equation for fluid dynamics and beyond", Oxford University press (2001).

[8] M. C. Sukop and D.T. Thorne, "Lattice Boltzmann modeling, An introduction for geoscientist and engineering", Springer. Heidelberg (2006).

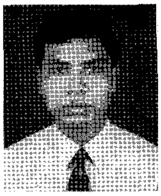
[9] M. Cheng, D.S. Whyte and J. Lou, "Numerical simulation of flow around a square cylinder in uniform- shear flow", J. Fluid and Structures, 23, pp. 207-226, 2007.

[10] O. Inoue, M. Mori and N. Hatakeyama, "Aeolian tones radiated from flow past two square cylinders in tandem", Physics of fluid, 18, pp. 046101-15, 2006.

[11] J.W.Kim, S.K.Oh and H.K.Kang, "Numerical investigation of flow-pattern and flow-induced noise for two staggered circular cylinder in cross-flow by LBM", J. of the KOSME, 28(1), pp. 82-93, 2008.

[12] E.R.Kim, J.H.Kim and H.K.Kang, "Numerical simulation of aeroacoustic noise at low mach number flows by using the finite difference Lattice Boltzmann method", J. of the KOSME, 28(5), pp.717-727, 2004.

Authors Profile

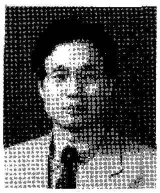


M. A. Taher

He Graduated (B. Sc. 1997 and M. Sc. 1998) in Applied Mathematics from the University of Rajshahi, Bangladesh. He was recruited as a lecturer of Mathematics Dept. in Dhaka University of Engineering & Technology (DUET), Gazipur-1700, Dhaka, Bangladesh since 2004. At present, he has been studying as a Ph.D. student, School of Mechanical Engineering (CFD & MEMS laboratory) in Pukyong National University, Busan, Korea.

Tae-Sil Baek

He garduated from Korea Maritime University(B.E.1983 and M.S. 1989) in Korea. He received his Ph.D degree from Korea Maritime University in 2001. He currently works at Pohang college as a professor



Yeon-Won Lee

He graduated from Kyungpook National University (B.E. 1981, M.S. 1983) in Korea. He received his Ph.D. degree from the University of Tokyo (Doctor of Engineering 1993) in Japan. He worked at POSCO E&C for 3 years as a Team Leader of Division of Mechanical Design. He visited the Institute of Industrial Science of the University of Tokyo as a visiting professor in 1997. He has been serving as a professor of the School of Mechanical Engineering in Pukyong National University since 1993. His research interests are CFD, various natural energy problems including wave energy conversion, and MEMS technology.