JOURNAL OF ENGINEERING AND TECHNOLOGY, Vol. 9 No. 1 June 2011 ISSN 1684-4114
http://www.iutoic-dhaka.edu/jet

Lattice-Boltzmann Analysis of Fluid Flow Behaviors around different Bluff Bodies in a 2D Micro-channel

M. A. Taher* and Y. W. Lee**

Received 14 May 2011; Accepted after revision 24 June 2011

ABSTRACT

In this study, the numerical analysis has been carried out to investigate the fluid flow behaviors around various bluff bodies in a two dimensional micro-channel using Lattice-Boltzmann Method (LBM).The LBM has been built up on the D2Q9 (two dimensional lattice with nine velocities) model with single relaxation method called Lattice-BGK (Bhatnagar-Gross-Krook) model. Streamlines, vorticity, velocity and pressure contours are provided to analyze the important characteristics of the flow field for a wide range of the non-dimensional parameter Reynolds number (Re).The simulation results are compared with the experimental results and the results obtained from the other numerical models and the agreement is found to be very reasonable and satisfactory.

Keywords: *Lattice Boltzmann, Particle Distribution Function, Bluff Body, Reynolds Number.*

* *Dept. of Mathematics, Dhaka University of Engineering &Technology, Gazipur, Bangladesh. Email: tahermath@ yahoo.com*
***School of Mechanical Engineering, Pukyong National University, Busan, S. Korea.*

1 INTRODUCTION

Lattice Boltzmann Method (LBM) has attracted much attention as a novel alternative to traditional methods for solving the Navier–Stokes (N-S) equations. In Computational Fluid Dynamics (CFD), fluid properties, such as density, pressure, velocity and temperature are typically described by the Navier-Stokes (N-S) equations, which have nonlinear terms making them too expensive to solve numerically. However, the LBM has demonstrated a significant potential and broad applicability with numerous computational advantages, such as the parallel of algorithm, the simplicity of programming, and the ability to incorporate microscopic interactions. It is commonly recognized that the LBM can faithfully be used to simulate the incompressible N-S equations with high accuracy and this lattice BGK (LBGK) model, the local equilibrium distribution has been chosen to recover the N-S macroscopic equations by different authors [1-3]. 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 [4]. 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 Mcnamar et al. [5]. Due to the sampling of the particle velocities around zero velocity, LBM is limited to the low Mach number (nearly incompressible flow) flow simulation [6]. Actually, LBM originated from the Lattice Gas Automata (LGA) method, which can be considered as a fictitious molecular dynamics (MD) in which space, time and particle velocities are all discrete. This discretization of space and time first proposed by Hardy et al. [7], was called HPP model, define on a square lattice for studying transport properties of fluid. When two microscopic particles arrive at a node in the two opposite direction, they immediately leave the node in the two others, previously unoccupied directions. These rules conserve mass and momentum. A historical important lattice gas model is the FHP model, introduced by Frisch et al. [8].The updating 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. A particularly simple linearized version of the collision operator makes use of a relaxation towards an equilibrium value using a single relaxation time parameter known as the so-called Bhatnagar-Gross-Krook (BGK) model first describe by Bhatnagar et al.[9]. The simulation of flow around bluff bodies both in numerical and experimental studied by [10-14]. In their study they consider very large Reynolds number and they used commercial tools. 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 authors [15-17]. The objective of this paper is to numerically study of fluid flow behaviors around bluff bodies using LBM where the flow can be driven with the pressure (density) gradient. The flow is very sensitive to the change of Reynolds number, a dimensionless parameter, based on the characteristic length of our bluff bodies, the maximum incoming flow velocity and also the nature of fluid transport properties.

2 FORMULATION OF THE PROBLEM

2.1 Mathematical Model

The Lattice Boltzmann method is based on the idea of lattice gas cellular automata (LGCA) to simulate the fluid motion by a simplified microscopic model in discrete time steps using a discrete phase space, i.e. discrete velocity and location. Each cell of the resulting lattice represents a volume element of the fluid, which consists of a collection of particles. Their motion is represented by a particle distribution function (PDF) at each grid point. The macroscopic variables of interest (i.e. density, pressure, velocity) can easily be obtained from these particle distribution functions (PDF). In contrast, the Boltzmann equation deals with the single particle distribution function (PDF), $f(\vec{r},\vec{\xi},t)$, where $\vec{\xi}$ denotes the particle velocity vector in phase space $(\vec{r},\vec{\xi})$ and time *t*. Neglecting external forces, the transport equation for *f* can be expressed by the Boltzmann equation (BE) as,

$$\frac{\partial f}{\partial t}+\left(\vec{\xi}.\vec{\nabla}\right)f=\Omega(f) \quad \dots \quad \dots \quad \dots \quad \dots \quad (1)$$

The terms on left hand side of **Eq.1** is called the transport term and the right hand side is called the collision term. A suitable linearized form of the collision operator for near equilibrium sate of low Mach number hydrodynamics is the single relaxation time approximation, also known as Bhatnagar – Gross -Krook (BGK) model.

$$\Omega_{BGK}(f)=-\frac{1}{\tau}(f-f^{eq}) \quad \dots \quad \dots \quad (2)$$

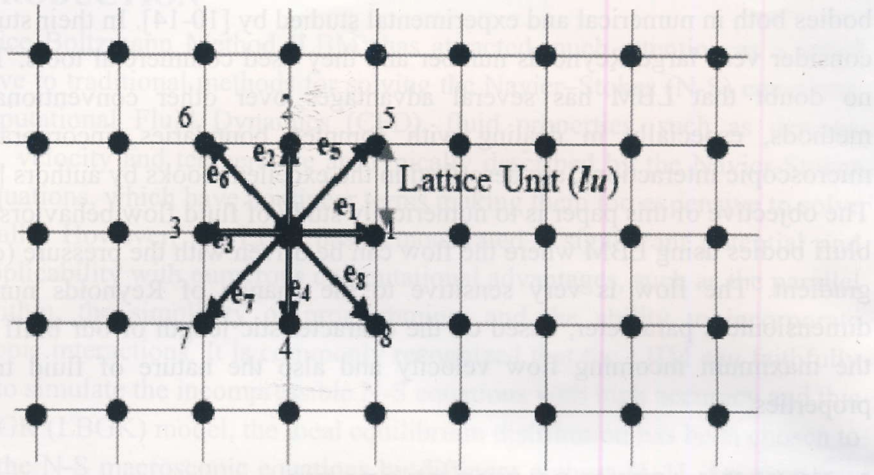


Figure 1: A schematic diagram of the D2Q9 square lattice model.

where, the relaxation parameter, τ, is linked to the viscosity (ν) of the fluid. In order to solve for f numerically, it is necessary to discretize the **Eq.1** in the velocity space using a finite set of particle velocity vectors $\vec{e}_i$, i=0, 1----, q-1, where q is the number of directions of the particle velocities at each node and then combining with **Eq.2**, the Boltzmann equation with BGK approximation becomes:

$$\frac{\partial f_i}{\partial t} + e_i \cdot \frac{\partial f_i}{\partial x_i} = -\frac{1}{\tau}(f_i - f_i^{eq}), \quad i = 0,1,2\text{-----}, q\text{-}1 \qquad \ldots \qquad (3)$$

Here, $f_i(\vec{x},t)$ is the velocity discrete particle distribution function and f_i^{eq} is the discrete equilibrium distribution function. It is almost impossible to perform the simulations with this form of the Boltzmann equation, if the phase (direction of particle velocities) is infinite. Hence, the simulation is carried out on a finite discrete phase space known as lattice and is labeled by DdQq with the number of space dimensions d and the number of discrete velocities q. In practice, there are three widely used 2D LBM geometries: D2Q5, D2Q7 and D2Q9. For D2Q9 model, each node of the lattice has three kinds of particle: a rest particle, particles that move in the co-ordinate directions and the particles that move in the diagonal directions as shown in **Fig.1**.

The total number of discrete velocities on each node in D2Q9 model is 9. The velocities of the particles are such that they move from one node to another during each time step. These particle velocities can be written as,

$$e_i = \begin{cases} (0,0) & i = 0, \\ c[\cos(\frac{i-1}{2}\pi), \sin(\frac{i-1}{2}\pi)] & i = 1,2,3,4, \\ \sqrt{2}c[\cos(\frac{i-5}{2}\pi + \frac{\pi}{4}), \sin(\frac{i-5}{2}\pi + \frac{\pi}{4})] & i = 5,6,7,8, \end{cases}$$

where, c is called the Courant-Friedrichs- Lewy (CFL) number. Therefore, the discrete form of **Eq.3** is called the Lattice Boltzmann equation (LBE) and can be written as,

$$f_i(\vec{x} + \Delta t\, \vec{e}_i, t + \Delta t) - f_i(\vec{x}, t) = -\frac{1}{\tau}(f_i - f_i^{eq}),\ \mathrm{i} = 0,1,2\text{-----}, 8 \quad \ldots \quad (4)$$

Here, $\omega = 1/\tau$ is the relaxation parameter. It also depends on the local macroscopic variables ρ and $\rho\vec{u}$ and satisfy the following laws of conservation:

$$\rho = \sum_i f_i^{eq} \text{ and } \rho\vec{u} = \sum_i \vec{e}_i f_i^{eq} \quad \ldots \quad \ldots \quad (5)$$

Note that the Navier –Stokes equations has a second order non-linearity. So the general form of the equilibrium distribution function can be written up to $O(u^2)$ [3],

$$f_i^{eq} = \rho w_i [1 + \frac{3}{c^2}\vec{e}_i.\vec{u} + \frac{9}{2c^4}(\vec{e}_i.\vec{u})^2 - \frac{3}{2c^2}u^2] \quad \ldots \quad (6)$$

where, the lattice weighting factor (w_i) depends only on the lattice model. For D2Q9 model, $w_0 = 4/9$, $w_i = 1/9$, $i = 1,2,3,4$, and $w_i = 1/36$, $i = 5,6,7,8$. Using the Chapman-Enskog expansion [6], it is mathematically provable that the **Eq.6** can recover the N-S equation to the limit of low Mach number if the pressure and the kinetic viscosity are defined by,

$$P = \rho Cs^2 \text{ and } \nu = (\tau\text{-}1/2)\, Cs^2 \Delta t \quad \ldots \quad (7)$$

Here, the speed of sound is defined by $Cs = \sqrt{RT}$. Here the temperature T has no physical significance as we are only dealing with the isothermal model (T = constant). And thus, the grid CFL number can be defined as $c = \sqrt{3RT} = \Delta x / \Delta t$. Therefore, the speed of sound becomes $Cs = c/\sqrt{3}$.

2.2 Numerical Analysis and Boundary Conditions

Consider two-dimensional steady compressible flow around a bluff body e.g. a square cylinder or a circular cylinder that placed symmetrically at the central line of the channel. The computational domain is to consider as a rectangular region $L \times H$, where L (= $4H$) is the length and H is the height of the channel as

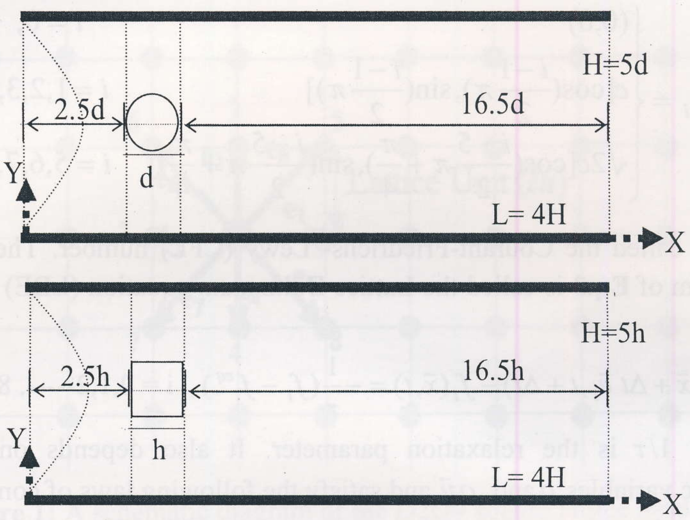


Figure 2: Physical models and coordinate systems.

shown in **Fig.2.** The upper and lower plates are stationary. The diameter (or h = height for square cylinder) of the circular cylinder is $d = H/5$. All simulations in this part are based on 400 × 100 uniform lattice nodes. The cylinder diameter (d) or height (h) is approximately 20 grid cells.

If the flow is laminar and fully developed, the velocity u is given by the Poiseuille–Hagen distribution,

$$u(y) = 4U\left(y/H - (y/H)^2\right) \quad \ldots \quad \ldots \quad \ldots \quad (8)$$

It is a parabolic velocity profile with the maximum velocity U at the center line of the channel. In an incompressible flow, the Reynolds number is one of the major controlling parameters that control the flow field. It is defined by $\mathrm{Re} = Uh/\nu$, where h is the characteristic length (or diameter d for circular cylinder) of the bluff body, U is the maximum incoming flow velocity (less than 0.1 *lu,* lattice unit) and ν is the kinematic viscosity of fluid. Actually, flow can be driven with the pressure (density) gradient of any desire magnitude by setting the boundaries densities in accordance with the **Eq.5** provided the maximum velocity remains small relative to 0.1 *lu*. Finally the **Eq.4** is solved on a uniform 2D grid system along with boundary conditions and other equations are described in section. Each numerical time steps consists of three stages:

(i) Collision,
(ii) Streaming, and
(iii) Boundary conditions followed by the LBM approaches.

Since the variables of interest are the macroscopic quantities, the LBM needs to be able to handle initial and boundary conditions that are prescribed in terms of the fluid density and velocity. The initial density is set to be 1.0 and the value of the velocity is set to be zero in the interior of the channel. The boundary conditions play an important role for the stability and the convergence of the Lattice-Boltzmann method. In the top and bottom wall, including the surfaces of the bluff body, no slip boundary conditions namely zero fluid velocity are applied. In LBM, there are two types of solid:

(i) Boundary solids that lie at the solid-fluid interface, called mid-grid or half way bounce-back no slip condition.

(ii) Isolated solids that do not contact fluid .i.e. the physical boundary lies exactly on a grid line, called on-grid or full bounce-back no slip condition or simply bounce back scheme.

The simulation has been performed by using LBM assumes the full way bounce back boundary conditions at upper and lower plate. In this rule; particles colliding with a wall simply reverse their velocities. For more details about boundary conditions, are given in reference [17]. Most LBM simulation Δx and Δt are assumed as the space and time unit respectively and it is called lattice unit. In this simulation, it is considered that, $\Delta x = 1lu = 1.27\times10^{-7}\ m$, $\Delta t = 1$, $ts = 4.17\times10^{-11}\ s$. The fluid properties are taken to water properties. The kinematic viscosity, $v = 1.006\times 10^{-6}\ m^2/s$ (water at 30^0C), corresponds to 0.004 lattice unit. All reported data are obtained on our calculation domain 400 × 100 (lattice node). Thus the physical domain of simulation is 40 μm × 10 μm. For accurate solution, the Mach number, Ma, should be kept as small as possible. In general, the maximum incoming fluid velocity U is considered in the LBM in order of 0.2 or 0.1 or less. Therefore, the Reynolds number should be chosen very carefully. The time is scaled with the lattice time unit and it is considered 100,000 time steps (iterations). Through out the simulation, the lattice units are considered. The computations were carried out with a code developed by the authors and written in FORTRAN language

3 RESULTS AND DISCUSSIONS

In conventional CFD methods for incompressible N-S equations, the Poisson equation is solved for the pressure, while in LBM, solving the **Eq.4** all information including pressure can obtain that are interested to our study. The numerical simulation of flow characteristic around a square cylinder and a circular cylinder were carried out for Reynolds number in the range up to 300.

Since, for this kind of flow, no experimental values are available, the typical features of the fluid flow behaviors are investigated. In order to assess the accuracy of this method, the result of LBM is compared with analytical solution define by,

$$u_{exact} = \frac{1}{2\rho\nu}(y^2 - yH)\frac{dp}{dx}, \quad \frac{dp}{dx} = -8\rho\nu U / H^2 \ldots \quad \ldots \quad (9)$$

Here H is the channel height and U is the maximum velocity located in the center of the channel.

In **Fig.3**, the result is compared with analytical solution for Re =100 and 200. The solid lines are the analytical solution and the dashes lines are the data results obtained from the simulation. This figure shows that the velocity profile in the channel is parabolic and the maximum value at the middle position of the channel. It is obviously as the fully developed laminar parabolic flow is considered and it is seen that our results (LMB) are in excellent agreement with analytical solution **Eq.9**. This confirms the accuracy of our simulation.

In **Fig.4**, the velocity in *x*-direction is plotted against the height of the channel at different locations x/L = 0.375 (near the body), 0.5 (cross section of the channel) and 0.75 (far from the body) for different values of Reynolds number. From **Figs.4** (i)-(iii), it is seen that the flow is almost symmetrical around the bluff bodies for low Reynolds number throughout the channel. It has two local maximum and one local minimum values, where the local maximum occur at the upper and lower part of the body and the minimum occur at the center part of the body. The solid lines represent the velocity for Re = 50 and dashes lines represent the velocity at Re = 100. However, the flow around

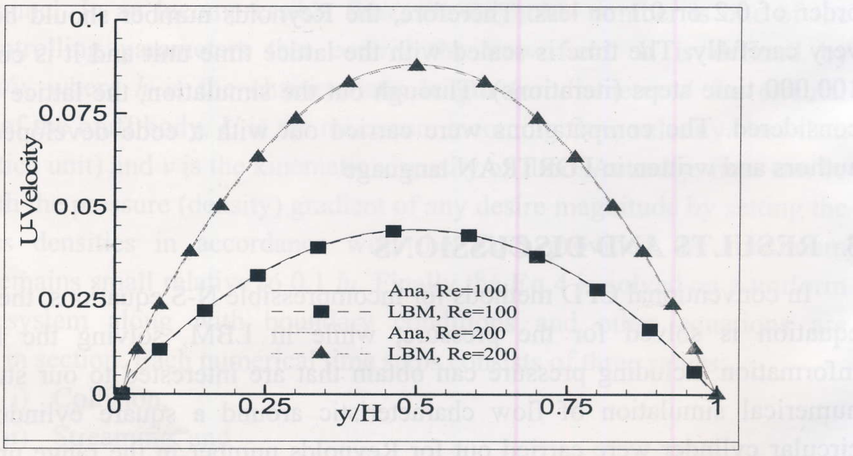


Figure 3: Verify LBM with analytical result for different values of Re.

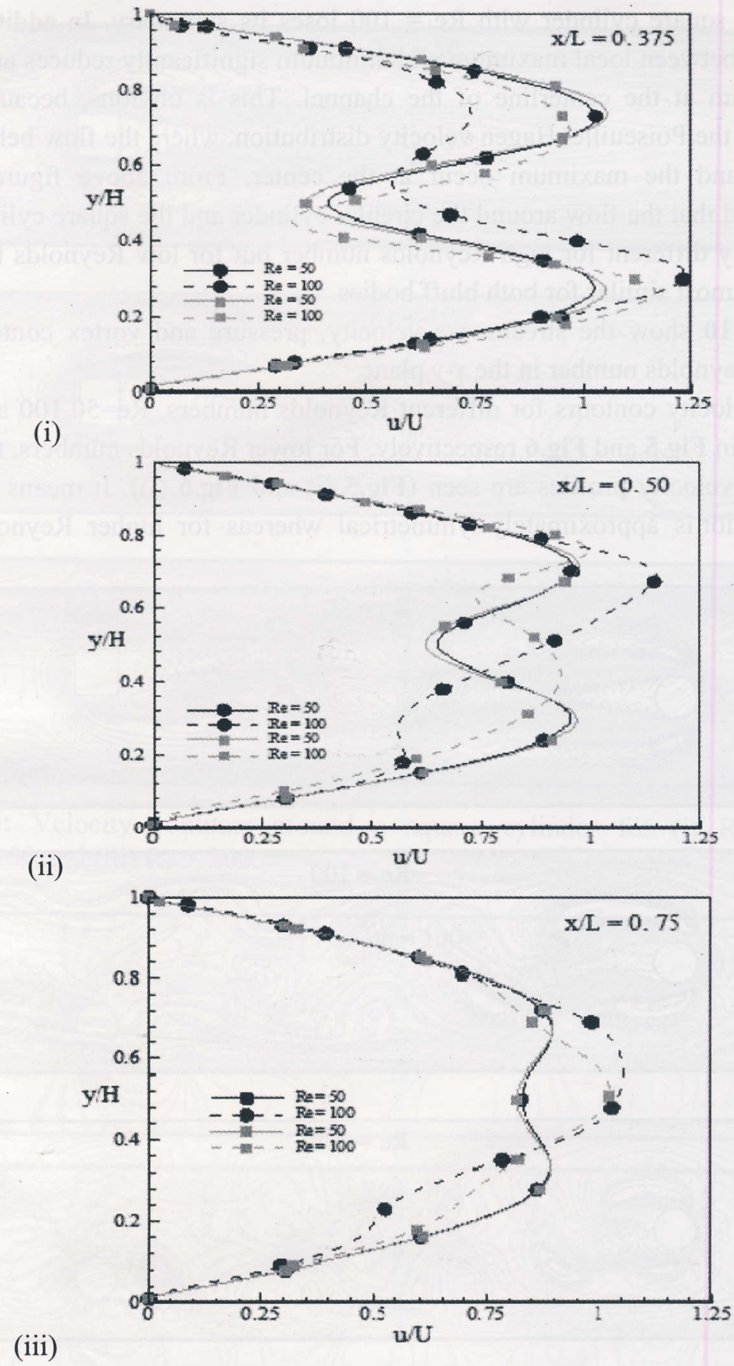


Figure 4: Velocity in *x*-direction around a circular cylinder (●) and a square cylinder (■) for different Re at (i) x/L=0.375, (ii) x/L=0.50 and (iii) x/L=0.75.

circular or square cylinder with Re = 100 loses its symmetry. In addition, the difference between local maximum and minimum significantly reduces and tends to maximum at the centerline of the channel. This is obvious, because, it is introduced the Poiseuille–Hagen velocity distribution, where the flow behavior is parabolic and the maximum occur at the center. From above figures, it is investigated that the flow around the circular cylinder and the square cylinder are significantly different for high Reynolds number but for low Reynolds the flow behavior almost similar for both bluff bodies.

Figs.5-10 show the streamwise velocity, pressure and vortex contours for different Reynolds number in the *x-y* plane.

The velocity contours for different Reynolds numbers, Re=50,100 and 200, are shown in **Fig.5** and **Fig.6** respectively. For lower Reynolds numbers, the fully developed velocity profiles are seen (**Fig.5** (i) and **Fig.6** (i)). It means that the flow behavior is approximately symmetrical whereas for higher Reynolds

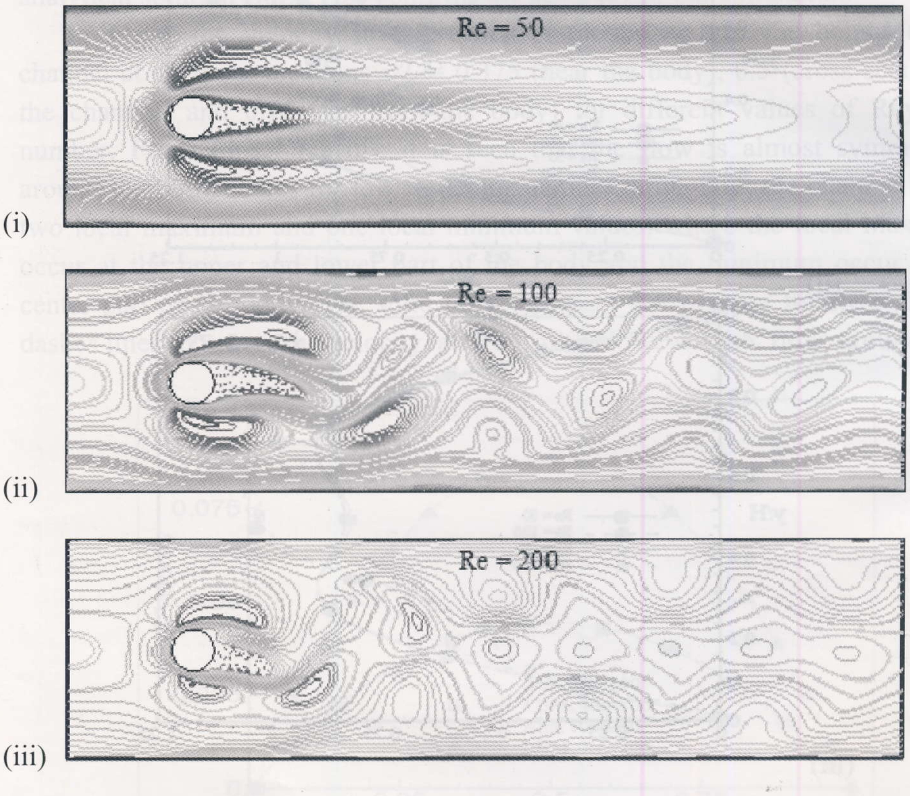


Figure 5: Velocity contours around a circular cylinder for (i) Re = 50, (ii) Re = 100 and (iii) Re = 200.

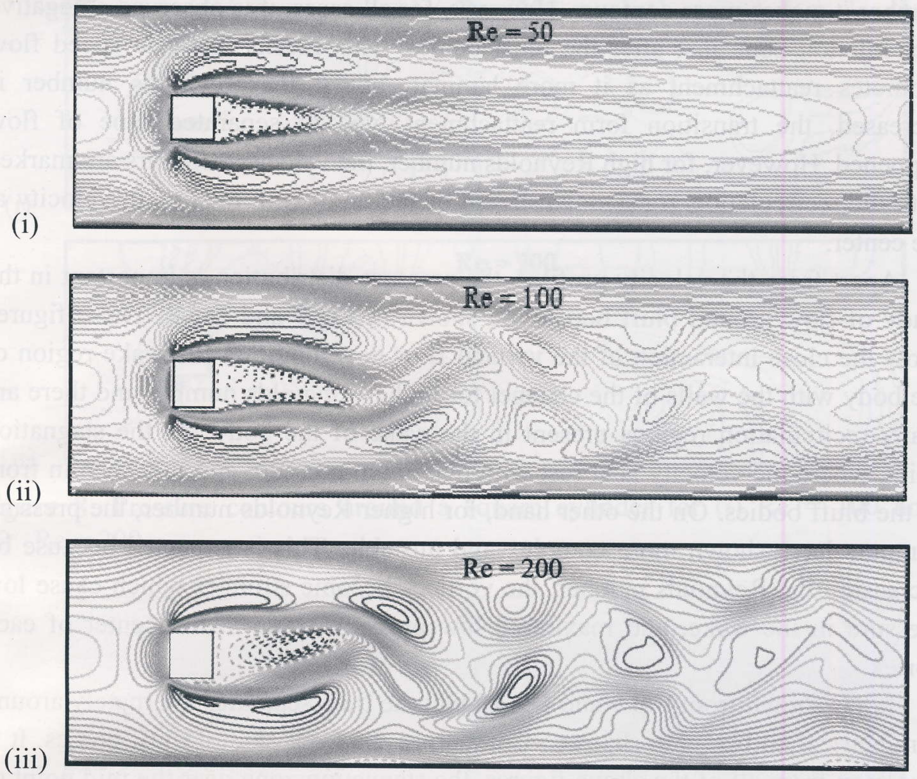


Figure 6: Velocity contours around a square cylinder for (i) Re = 50, (ii) Re = 100 and (iii) Re = 200.

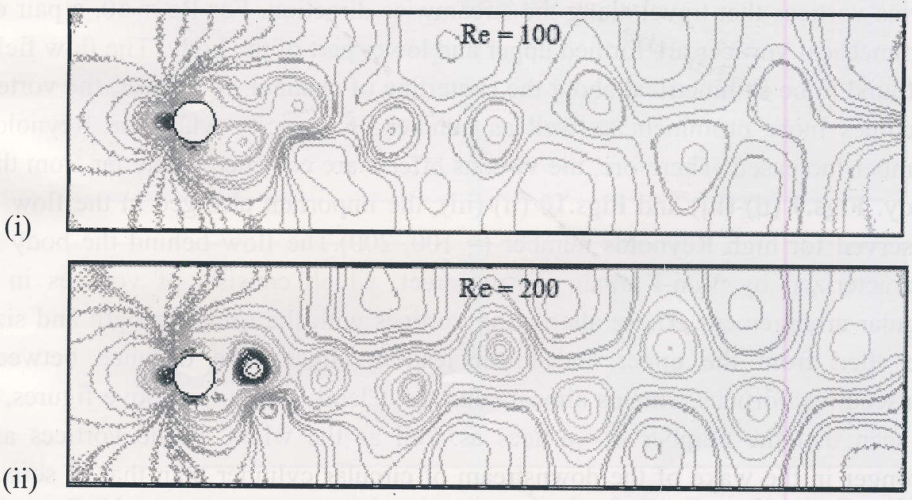
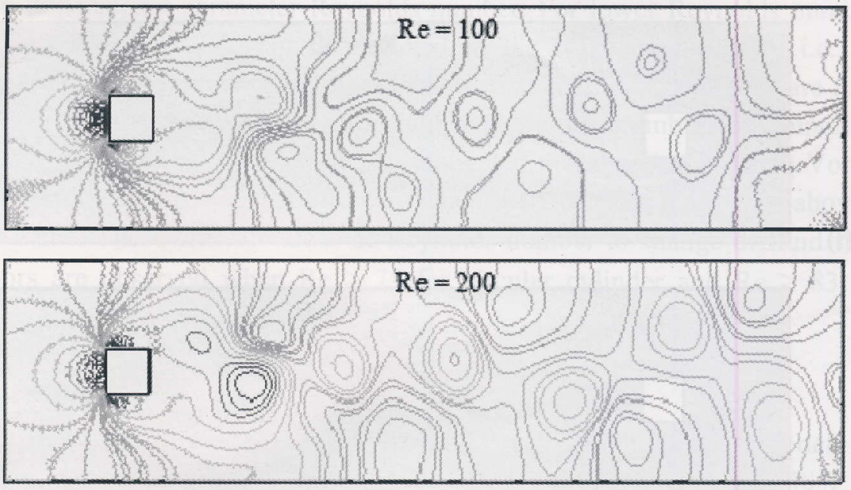


Figure 7: Pressure contours around a circular cylinder for (i) Re = 100 and (ii) Re = 200.

number it makes more contours. However, for all cases, it is observed a negative recirculation zone just after the bluff bodies. This means that low speed flow promotes reattachment as it more laminar, but if the Reynolds number is increased, the transition form reattachment type to separated type of flow happened. However, for high Reynolds number, the velocity contours are marked with the recirculation zones downstream of the body with maximum velocity at the center.

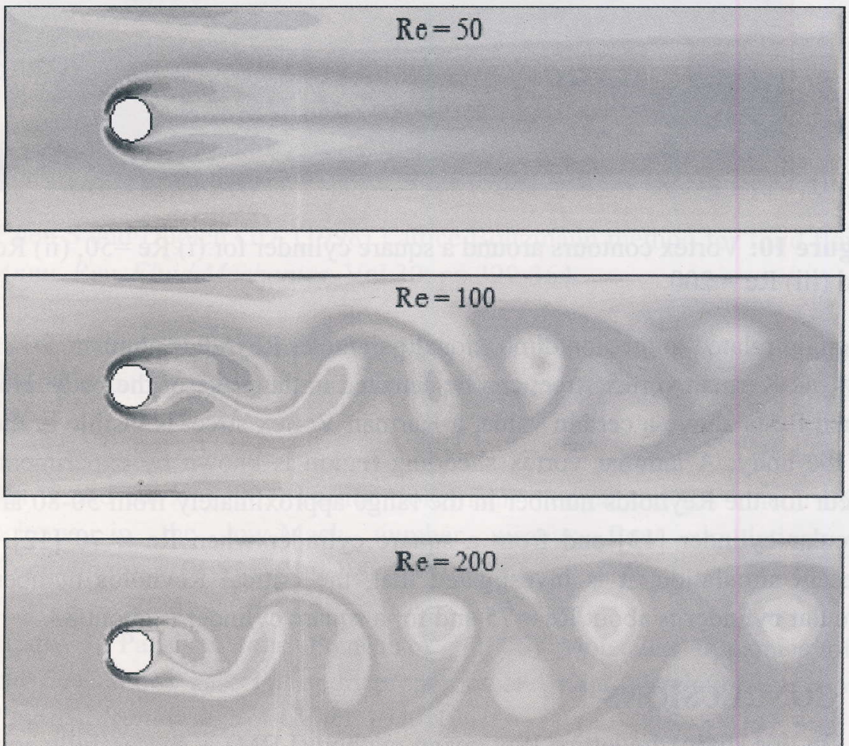
Apart from the velocity profiles, the pressure distribution is important in the study of flow around bluff bodies are shown in **Fig.7** and **Fig.8.** These figures show the close interaction of the vertical flow developing in the wake region of the body with the walls of the channel for higher Reynolds number and there are many recirculation regions present in the wake of the body. At the stagnation point, where the velocity is at rest with maximum pressure and it is seen in front of the bluff bodies. On the other hand, for higher Reynolds number, the pressure contours have shown more complex and unstable. This is expected because by increasing the Reynolds number, the vortices become stronger which cause low pressure in the wake, and maximum low pressure attain at the center of each vortex.

It has shown from **Fig.9** and **Fig.10** that the flow separates alternately around symmetrical bodies with sharp corner such as the front side of bluff bodies. It is also seen from all of the above figures, the stagnation zone near the mid point of the front face of the cylinders, where the flow divided equally into two parts. Since the flow separates from the leading edges corner of the bluff bodies, it forms vortices that travel along the streamwise direction. For Re = 50, a pair of symmetrical vortices are formed upper and lower part of the body. The flow field is found to be symmetrical about the centerline of channel. Moreover, the vortex becomes more prominent as well as number of vorticity when the Reynolds number increased. Therefore, the viscous effects are confined to the far from the body. **Figs.9** (ii)-(iii) and **Figs.10** (ii)-(iii), the important changed in the flow is observed for high Reynolds number (= 100, 200).The flow behind the body is characterized by Von-Karman vortex street, which consists of vortices in a regular arrangement. These alternating vortices with the same strength and size are shed from the upper and lower leading edges. The distance between consecutive vortices remains almost constant. Clearly, from the above figures, it is seen that the number of vortices as well as the width of the vortices are stronger in the wake of the downstream of circular cylinder than that of square cylinder. It is obvious that the flow pattern in the wake of the bluff body is



(i) (ii)

Figure 8: Pressure contours around a square cylinder for (i) Re = 100 and (ii) Re = 200.



(i) (ii) (iii)

Figure 9: Vortex contours around a circular cylinder for (i) Re = 50, (ii) Re =100 and (iii) Re = 200.

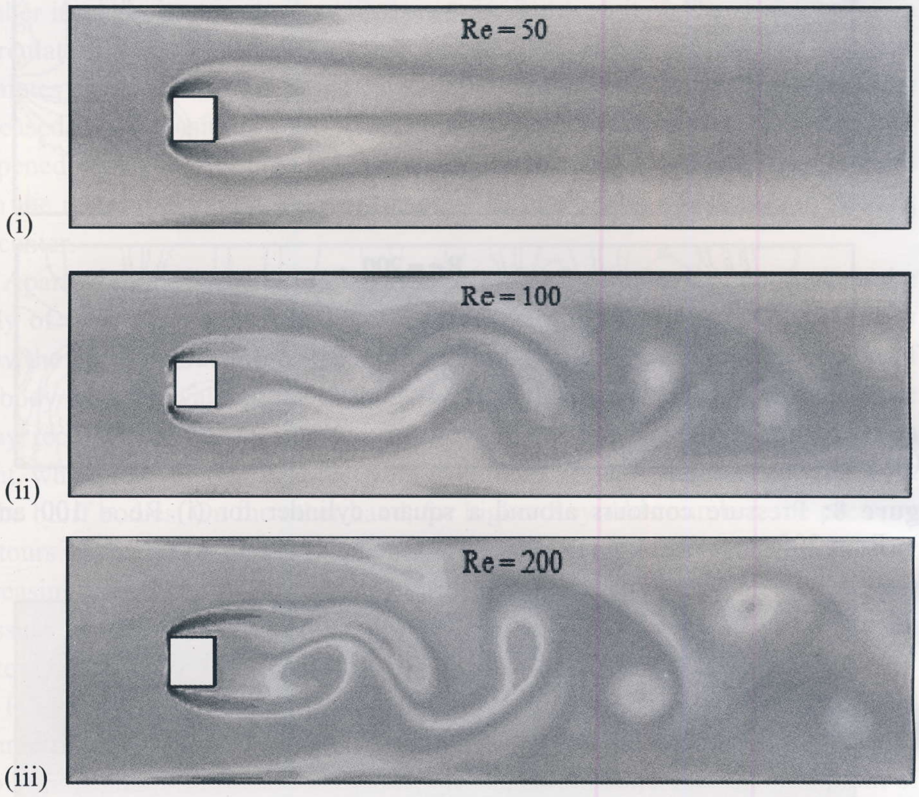


Figure 10: Vortex contours around a square cylinder for (i) Re =50, (ii) Re =100 and (iii) Re = 200.

strongly related to the non dimensional parameter Reynolds number. When Re is low, no Karman vortex street can be detected in the wake of the body. However, when Re is above a certain value, a Karman Vortex street is visible in the wake of the body. A laminar vortex shedding region is known by experimentally to occur for the Reynolds number in the range approximately from 50-80 around a circular cylinder [13] and from a square cylinder when Re = 70 [12]. In the present simulation, it is investigated that, the critical Reynolds number for a circular cylinder is about Re = 75 and for a square cylinder is about 83.

4 CONCLUSIONS

In this study, the lattice Boltzmann method (LBM) is successfully applied to simulate a two-dimensional channel flow driven by a pressure difference around bluff bodies. The flow pattern in the wake of the bluff body is strongly related to the non dimensional parameter Reynolds number. For lower Reynolds number, no Karman vortex street can be detected in the wake of the body i.e. the variations in the contours are not significant as the recirculation regions are fully developed. However, in the case of higher Reynolds number, an important changed in the flow is observed. It is investigated that the Von Karman Vortex street is visible in the wake of the body when the Reynolds number is above a certain value. These critical values of Reynolds number to change of fluid flow behaviors are observed when Re $\geq$ 75 for circular cylinder and Re $\geq$ 83 for square cylinder.

REFERENCES

[1] Qian YH, D'Humieres D and Lallemand P. (1992) Lattice BGK models for Navier-Stokes equation. *Euro physics letter*, Vol.17, No.6: pp.479-484.

[2] Zou Q and He X. (1997) On pressure and velocity boundary conditions for the lattice Boltzmann BGK model. *Phys. Fluids*, Vol.9, No.6: pp.1591-1598.

[3] Chen H, Chen S and Matthaeus WH. (1992) Recovery of the Navier-Stokes equations using a lattice-gas Boltzmann method. *Physical Review A*, Vol.45, No.8: pp.5339-5342.

[4] Chen S and Doolen GD. (1998) Lattice Boltzmann method for fluid flows. *Annu. Rev. Fluid Mechanics,* Vol.30: pp.329-364.

[5] McNamara GR and Zanetti G. (1998) Use of the Boltzmann equation to simulate lattice-gas automata. *Physical review letters,* Vol.61: pp.2332-2335.

[6] Xu K and He X. (2003) Lattice Boltzmann method and gas kinetic BGK scheme in the low-Mach number viscous flow simulations. *J. Computational Physics*, Vol.190: pp.100-117.

[7] Hardy J, Pazzis O and Pomeau Y. (1976) Molecular dynamics of a classical lattice gas: Transport properties and time correlation functions. *Physical Review A*, Vol.13: pp. 1949-1961.

[8] Frishch U, Hasslacher B and Pomeau Y. (1986) Lattice-gas automata for the Navier-Stokes equation. *Physical Review Letters*, Vol.56: pp.1505-1508.

[9] Bhatnangar PL, Gross EP and Krook M. (1954) A model for collision processes in gases. I.Small amplitude processes in charged and neutral one-component system. *Physical Review Letters*, Vol.94, No.3: pp.511-525.

[10] Shiels D. (1998) Simulation of controlled bluff body flow with a viscous vortex method. *Ph. D thesis, California Institute of technology, California.*

[11] Taher MA, Baek T-S and Lee YW. (2009) Lattice-Boltzmann simulation of fluid flow around a pair of rectangular cylinders. *J. Korean Society of Marine Engineering*, Vol. 33, No.01: pp.62-70.

[12] Breuer M, Bernsdorf J, Zeiser T and Durst F. (2000) Accurate computations of the laminar flow past a square cylinder based on two different methods: lattice Boltzmann and finite volume. *Int. J. Heat and Fluid Flow*, Vol.21: pp.186-196.

[13] Williamson CHK. (1998) Defining a universal and continuous Strouhal-Reynolds number relationship for a laminar vortex shedding of a circular cylinder. *Physics of Fluid*, Vol.31, No.10: pp.2742-2744.

[14] Liaw KF. (2005) Simulation of flow around bluff bodies and bridge deck Sections using CFD. *Ph. D Thesis, University of Nottingham, Nottingham.*

[15] Succi S. (2001) The lattice Boltzmann equation for fluid dynamics and beyond. *Oxford University press, New York.*

[16] Suukop MC and Thorne DT. (2006) Lattice Boltzmann modeling, An introduction for geoscientist and engineering. *Springer, Heidelberg.*