Numerical study of the effect of wedge angle of a wedge-shaped body in a channel using lattice Boltzmann method

M.A. Taher\textsuperscript{a*}, Y.W. Lee\textsuperscript{b}, H.D. Kim\textsuperscript{c}

\textsuperscript{a}Department of Mathematics, Dhaka University of Engineering & Technology, Gazipur, Bangladesh
\textsuperscript{b}School of Mechanical Engineering, Pukyong National University, Busan, Korea
\textsuperscript{c}School of Mechanical Engineering, Andong National University, Andong, Korea

Abstract

The aim of this paper is to study the fluid flow behavior around a wedge shaped body with different wedge angles placed in a channel 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. The influence of the gap ratio $G^*=G/H$, where $H$ is the distance between two parallel walls, $G$ is the gap between body and wall, on the flow field is illustrated. The gap ratio, $G^*$, depends on the angle of wedge-shaped body ($0^\circ < \theta < 180^\circ$). Streamlines, vorticity contours and pressure contours are provided to analyze the important characteristics of the flow field for a wide range of non dimensional parameters namely the Reynolds number (Re), Strouhal number (St) and the gap ratio($G^*$). However, it is seen that the flow can be characterized by three regions: (i) large gap ratio, $0.44 < G^* < 0.50$, with $0^\circ < \theta < 55^\circ$, (ii) intermediate gap ratio, $0.20 \leq G^* \leq 0.44$, with $55^\circ \leq \theta \leq 120^\circ$ and (iii) small gap ratio, $0 < G^* < 0.20$, with $120^\circ < \theta < 180^\circ$. The simulation results are compared with experimental data and other numerical models and found to be very reasonable and satisfactory.

Keywords: Lattice-Boltzmann; Bhatnagar-Gross-Krook; Reynolds number; Strouhal number; wedge-shaped, gap ratio.

Nomenclature

$C_p$ Specfic heat at constant pressure (J.kg\textsuperscript{-2} K\textsuperscript{-1})
$C_s$ Speed of sound (m.s\textsuperscript{-1})
c CFL number
e\textsubscript{i} Discrete particle velocity vector (m s\textsuperscript{-1})
$F_i$ Discrete particle distribution function
$F_{eq}\textsuperscript{i}$ Discrete particle distribution function
$f$ Shedding frequency
$G$ Gap between body and wall (m)
$G^*$ Gap ratio
$H$ Channel height (m)
$L$ Channel length (m)
$Re$ Reynolds number
$St$ Strouhal number
$U$ Characteristic velocity (m.s\textsuperscript{-1})
1. Introduction

In the past years, the Lattice Boltzmann Method (LBM) has attracted much attention as a novel alternative to traditional computational fluid dynamics (CFD) methods for numerically solving the Navier–Stokes (N-S) equations. Actually the 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. 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 have been discussed by McNamara et al. [1] and Wei et al. [2]. Due to the sampling of the particle velocities around zero velocity, LBM is limited to the low Mach number (nearly incompressible flow) flow simulation. It is commonly recognized that the LBM can faithfully be used to simulated the incompressible Navier-Stokes (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 [3-5]. It is found that the simulation results from LBM are in good quantitative agreement with experimental results. However, He and Luo [6] shown that the lattice Boltzmann equation is directly derived form the Boltzmann equation with various approximations by discretization in both space and time. In their study, they demonstrated that simulation results from LBM are in good quantitative agreement with experimental results. 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 [7]. Taher and Lee [8] have investigated numerically the suppression of fluid forces acting on a bluff body with different control bodies. It is found that the fluid forces acting on the main bluff body are effectively suppressed if the control body (a thin plate or a small circular cylinder) is placed at a suitable position with proper height or diameter. Moreover, LBM has several advantages over other conventional CFD methods, especially in dealing with complex boundaries, incorporating of microscopic interactions, and parallelization of the algorithm that are described in the excellent books by authors [9-12]. The viscous flow past a bluff body and the resulting separated region behind it has been focus on numerous experimental and numerical investigations. There is no doubt that an enormous corpus of literature on the subject of bluff body wakes has developed since the pioneering work of Strouhal and Von Karman. This flow situation is popular not only because of its academic attractiveness but also owing to its related technical problems associated with energy conservation and structural design. This type of flow is of relevance for many practical applications, e.g., vortex flow meter, buildings, bridge, towers, masts and wires. A laminar vortex shedding region is known to occur for the Reynolds number range extending approximately from 50-80 and the universal relationship between Reynolds and Strouhal numbers around a circular cylinder have been studied by and Williamson [13]. Actually many authors have been studied the vortex shedding frequency behind a circular cylinder or square cylinder or two cylinders for different cases both in numerically and experimentally However, in this paper, the present authors want to study the fluid flow behaviors around wedge-shaped using lattice Boltzmann method (LBM). As far we know, the problem has not been considered before. The objective of this paper is to numerically study of fluid flow behavior around wedge-shaped body using LBM where flow can be driven with the pressure (density) gradients. Computations are carried out for various wedge angles ranging from 0° to 180° and the Reynolds number ranging from 0 to 397, based on the characteristic length of the channel, the maximum incoming flow velocity (less than 0.1 lu) and also the nature of fluid transport properties. Here we have focused our attention on the evolution of streamlines, vorticity contours, pressure contours as well as velocity profiles and vortex shedding frequency, to investigate the important characteristics of the flow field around wed-
shaped body for a wide range of non-dimensional parameters that present in our simulation namely Reynolds number \( (Re) \), Strouhal number \( (St) \) and Gap ratio \( (G^*) \). Throughout our calculation, we use Lattice-Boltzmann units.

2. Formulation of the problem

The computational domain is to consider as a rectangular region \( L\times H \), where \( H \) is the height and \( L=4H \) is the length of the channel. A wedge-shaped body having wedge angle \( \beta \) placed symmetrically between parallel walls as shown in Fig.1

![Fig.1. Physical model and coordinate systems.](image)

The wedge angle is defined by \( \theta = \pi \beta \), \( 0 \leq \beta \leq 1 \). It is noted that, in the case of \( \beta \) equal to zero corresponds to flow over a horizontal flat plate while \( \beta \) equal to 1.0 corresponds to flow over a vertical flat plate. So the angle measurement is very important in this analysis. If \( G \) is the gap between wall and the body then \( G^* = G/H \) is defined as a gap ratio. It is noted that if the value of \( \beta \) increase, then \( G^* \) decreases. For convenient, it has been discussed in the present study of the following three different cases.

<table>
<thead>
<tr>
<th>Case</th>
<th>( \beta )</th>
<th>( \theta )</th>
<th>( G^* ) (=( G/H ))</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>0.25</td>
<td>45(^{\circ})</td>
<td>0.45</td>
</tr>
<tr>
<td>Case 2</td>
<td>0.50</td>
<td>90(^{\circ})</td>
<td>0.375</td>
</tr>
<tr>
<td>Case 3</td>
<td>0.75</td>
<td>135(^{\circ})</td>
<td>0.20</td>
</tr>
</tbody>
</table>

In an incompressible flow, Reynolds number is the only parameter that controls the flow field and is define by \( Re = \frac{UD}{\nu} \), where \( U \) and \( D \) are the characteristic velocity and the length respectively. In fluid dynamics, vorticity is the circulation per unit area at a point in the flow field. Mathematically, it is defined as, \( \vec{\omega} = \nabla \times \vec{u} \), where \( \vec{u} \) is the fluid velocity. The non dimensional shedding frequency, the Strouhal number, is defined as: \( St = \frac{fD}{U} \), where \( f \) is the vortex shedding frequency. This relation is believed to be accurate to ±1% in the Reynolds number. In this section, it is assumed that, \( \Delta x = \frac{1}{10} = 1.27 \times 10^{-6} \text{ m} \), \( \Delta t = 1 \times 10^{-9} \text{ s} \). The fluid properties are taken to air properties. The kinematic viscosity \( \nu = 15.636 \times 10^{-6} \text{ m}^2/\text{s} \), which corresponds to 0.024 lattice unit. All reported data are obtained on our calculation domain 320x80 (lattice node). Thus the physical domain of simulation is 400 \( \mu \text{m} \times 100 \mu \text{m} \). 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.

In order to simulate a fully developed laminar channel flow upstream of the wedge-shaped body, a parabolic velocity profile can be used with a maximum velocity \( U \) at the midpoint of the channel. This velocity is chosen to be lower than 10% of the speed of sound for LBM simulations to avoid significant compressibility effects. In our simulation, we use Zou-He Boundary condition to implement Dirichlet boundaries on inlet/outlet. At the top and bottom wall, no slip boundary conditions were imposed by the standard bounce back treatment. In LBM, the movement of the fluid particles is modeled instead of directly solving the macroscopic fluid quantities like the velocity and the pressure. It is known as mesoscopic simulation model, which is based on the Boltzmann equation. Neglecting external forces, the Boltzmann equation (BE) with BGK approximation can be written as

\[
\frac{\partial F_i}{\partial t} + \vec{c}_i \cdot \frac{\partial F_i}{\partial \vec{x}} = -\frac{1}{\tau} (F_i - F_i^{eq}) , \quad i = 0,1,2,3,........, q-1
\]

Where, \( F_i(\vec{x}, t) \) is the discrete particle distribution function and \( F_i^{eq} \) is the discrete equilibrium distribution function at lattice position \( \vec{x} \) and time \( t \) defined by
The lattice weighting factors, $w_i$, depend on the lattice model. For D2Q9 model, each node of the lattice has three kinds of particle: a rest particle that resides in the node, particles that move in the co-ordinate directions and the particles that move in the diagonal directions. So the total number of discrete velocities ($e_i$) on each node in D2Q9 model is 9.

Table 2. D2Q9 lattice velocities.

| $e_0 = (0, 0)$ | $e_1 = (c, 0)$ | $e_2 = (0, c)$ | $e_3 = (-c, 0)$ | $e_4 = (0, -c)$ |
| $e_5 = (c, c)$ | $e_6 = (-c, c)$ | $e_7 = (-c, -c)$ | $e_8 = (-c, c)$ |

Here $c$ is called the Courant-Friedrichs- Lewy (CFL) number and is proportional to $\Delta x/\Delta t$, where $\Delta x$ and $\Delta t$ are the lattice space and the time steps respectively. Therefore the discrete form of equation (1) is called the Lattice-Boltzmann equation (LBE) and can be defined as

$$F_i(\bar{x} + \Delta t \bar{e}_i, t + \Delta t) - F_i(\bar{x}, t) = \frac{1}{\tau} (F_i - F_i^{eq}), \quad i = 0, 1, 2, \ldots, 8$$

(3)

The relaxation parameter, $\omega = 1/\tau$, depends on the local macroscopic variables $\rho$ and $\rho \bar{u}$. These variables should satisfy the following laws of conservation:

$$\rho = \sum_i F_i, \quad \rho \bar{u} = \sum_i \bar{e}_i F_i$$

(4)

The above expressions describe the relationships between the microscaled quantities and the macro scaled physical quantities. Using the Chapman-Enskog expansion, i.e. multi-scale analysis, it is mathematically provable that the LBM equation (3) can recover the N-S equation to the second order of accuracy in the limit of low Mach number [5], if the pressure and the kinetic viscosity are defined by $P = \rho C_s^2$ and $\nu = (\tau/12) C_s^2 \Delta t$.

3. Results and discussions

In this problem, equation (3) is an algebraic equation. 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 information that we interested to our study. In Fig.2, we compare our result with analytical solution for Re =100, 200 in a channel in order to assess the accuracy of our method.

![Fig.2. Verify LB with analytical result for different Reynolds numbers](image_url)

The solid lines are the analytical solution and the dashed 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 we consider the fully developed laminar parabolic flow and it is seen that our results are in excellent agreement with analytical solution. This confirms the accuracy of our present simulation. One important quantity taken into account in the present study is the Strouhal number($St = fh/U$), computed from the height ($h$) of the bluff body,
the vortex shedding frequency \((f)\) and the velocity of the incoming fluid. The dimensionless shedding frequency with Reynolds number along the wake centerline downstream of the wedge-shaped body is shown in Fig.3

![Variation of dimensionless shedding frequency with Reynolds number](image)

**Fig.3.** Variation of dimensionless shedding frequency with Reynolds number

It is noted that the flow velocity profile, the position, shape of the bluff (barrier) and the ratio of the cross section area of the bluff to the wall affect the Strouhal number \((St)\) for the given Reynolds number \((Re)\) of the flow regime. As the dimensionless vortex frequency increases when the gap ratio decreases with the Reynolds number in the range 45-275 and consequently the Strouhal number also increases within this range. The higher frequency means that the process of vortex shedding is faster. Therefore, the nature of the vortex shedding is a strong function of the Re. In order to gain further insight into the evolution of vortex shedding in the near-weak region, the patterns of the vorticity contours for \(Re=100, 200\) and \(300\) with different gap ratios are plotted in Figs. 4-6.

![Vorticity distribution for \(\theta=45^\circ\) with \(Re\)](image)

**Fig.4.** Vorticity distribution for \(\theta=45^\circ\) with \(Re\)

![Vorticity distribution for \(\theta=90^\circ\) with \(Re\)](image)

**Fig.5.** Vorticity distribution for \(\theta=90^\circ\) with \(Re\)

![Vorticity distribution for \(\theta=135^\circ\) with \(Re\)](image)

**Fig.6.** Vorticity distribution for \(\theta=135^\circ\) with \(Re\)

For small wedge angle, higher gap ratio, it is seen that a pair of vortices with same strength and size are formed behind the body; a positive vortex (anticlockwise) appears on the lower part of the body and a negative vortex (clockwise) on the
upper part of the body. Further, it is observed that for low wedge angles ($<55^\circ$), the flow patterns are almost symmetrical for all Reynolds number within the range up to 300. However, for large wedge angles, an unsymmetrical flow patterns have been seen around the body as shown in the Fig.5-6. For, $\theta = 90^\circ$ with $Re < 195$, the fluid flow behaviors is almost symmetric. Moreover, a significant changed in the flow is observed when $Re \geq 195$. In Fig. 5(b)-(c), the Von Karman Street is seen, which is consists of vortices in a regular arrangement. For $Re=200$, the width of the vortex street behind the body is narrower compare to the $Re = 300$. The width is increasing with the increasing of Reynolds number. When $Re \geq 230$, the strong vortices are observed i.e. the width of the vortex street and the numbers of vortices are remarkably changed. There are seven vortices are seen in Fig.5(c) within the considered wake region for $Re=300$. This kind of flow behaviors are observed with the wedge angles in the range of $55^\circ \leq \theta \leq 120^\circ$. Further, the increases in the wedge angle (Fig.6) correspond to very low gap ratio, a pair of vortices are formed just behind the body, and in addition, the wall proximity effects are seen to give rise to reverse Von Karman Street.

A detailed view of flow field behind the wedge-shaped body and changes in the vortex shedding pattern with different time steps are shown in Fig.7. The flow is developed until 20,000 time steps. In Fig.7 (a), a new vortex is forming on the top of the body but the lower one is pulled away from the body. The formation of the upper vortex is completed at 20,500 time steps and consequently another new vortex on the bottom is forming and it is observed that the vortices are shed alternately with different time steps. Finally, the last plot, at time steps $t = 21500$, is nearly identical with the Fig.7 (a). This evolutionary process is repeated approximately every 1500 time steps. This time period is strongly depends on the Reynolds number. If the Reynolds number increases, the time period becomes shorter. It is investigated that for $Re=300$, the time period is approximately 1000 time steps. The same phenomenon has been seen that for flow over an airfoil at -90 degrees angles of attack documented by Rogers and Kwak [14].

Typical examples of instantaneous flow fields are presented in terms of vorticity for various time steps with wedge angle, $\theta =135^\circ$, corresponding to the gap ratio $G^* = 0.20$. This is easily understood by examining streamlines shown in Figs.8 (a)-(c). At $t = 5,000$ time steps, two opposite vortices with almost similar size and shape are seen just behind the body and simultaneously three stationary vortices are seen on the top wall whereas two vortices are seen on the bottom wall of the channel. The recirculation regions on the bottom wall are much larger than those of top wall. In Fig.8 (b), the upper vortex just behind the body a little extended and the two stationary vortices on the top wall far from the body are merging but the one (near the body) is still observed. However, the two vortices at the bottom wall have shown to tendency to become a large one. Finally, at $t = 15,000$, the two vortices on the bottom wall converted to one big size vortex with same center. After $t =15,000$ times steps, there is no changed of the flow field. Thus it is concluded that, for higher wedge angle, small gap ratios, the change of flow field occurred until approximately 15,000 time steps. However, the flow separation on the wall is observed in this case. This phenomenon is observed when $Re \geq 130$. 

![Fig.7. Streamlines plot for different time steps with $Re = 260$ and $\theta = 90^\circ$.](image1)

![Fig.8. Streamlines plot for different time steps with $Re=300$, $\theta=135^\circ$.](image2)
Apart from the velocity profile, pressure distribution is important to understand the flow field behavior around the bluff body. Fig. 8 shows the pressure contours with Reynolds number $Re=300$ for different angles $\theta = 45^0$, $90^0$ and $135^0$. In all cases, the pressure contour at the frontal stagnation point has maximum value where as just behind the body has minimum. Fig. 9 (a), for large gap ratio, the variations of the pressure contours are not significant because the recirculation regions as well as the flow in the wake are fully developed. However, if the wedge angle increases, Figs. 9 (b), the pressure contours becomes more complicated patterns and many recirculation regions are observed. The pressure contours indicate the location of vortex contours, where the pressure has a local minimum value. Further increased the wedge angle, Fig. 9(c), corresponds to very low gap ratio, the wall effects are considerable. Therefore, there exist a little recirculation region and the variation of pressure contours are not significant at the far from the body. The same characteristic has found in Fig. 6.

4. Conclusions

In this study, the flow can be characterized by three regions: (i) large gap ratio, $0.44 < G^* < 0.50$, with $0^0 < \theta < 55^0$, (ii) intermediate gap ratio, $0.20 \leq G^* \leq 0.44$, with $55^0 \leq \theta \leq 120^0$ and (iii) small gap ratio, $0 < G^* < 0.20$, with $120^0 < \theta < 180^0$.

The investigations of these regions are as follows:

- For case 1, $\theta = 45^0$, the flow is almost symmetrical.
- For case 2, $\theta = 90^0$, the flow behind the body is characterized by a Karman vortex street when $Re \geq 195$ and the vortices become stronger with increasing the Reynolds number. The formation and the shedding of vortices are repeated during a time period. This time period becomes shorter with increasing the Reynolds numbers. For $Re=260$, the time period is approximately 1500 times steps (lattice unit) where as for $Re=300$, it is seen approximately 1000 time steps. This kind of flow behaviors are observed with the wedge angles in the range of $55^0 < \theta < 120^0$.
- For case 3, $\theta = 135^0$, the wall proximity effects are observed to give rise to reverse Von Karman street and consequently a packet of vortices are created on the both channel walls and it is observed when $Re \geq 130$.

References