Numerical Simulation of 2-D Incompressible Flow in Micro Semi-circular Cavity by LBM

M. Alafzadeh*
Department of Mechanical Engineering,
Isfahan University of Technology, Iran
Email: m.alafzadeh@me.iut.ac.ir
*Corresponding author

Sh. Talebi
Department of Mechanical Engineering,
Yazd University, Iran
Email: talebi_s@yazduni.ac.ir

Received 10 November 2011; Revised 25 December 2011; Accepted 18 February 2012

Abstract: Since the field of micro electromechanical systems (MEMS) continues to grow, the flow in micro devices have become an area that receives a significant attention. Microscopic flows are usually characterized by the Knudsen number (Kn). When the characteristic size of the particle decreases down to a value comparable to the mean free path of the molecules, the continuum assumption fails and the Navier-stokes equations with No-slip boundary conditions cannot be applied, so the numerical method which is not based on continuity of the flow is needed. Recently the Lattice Boltzmann Method (LBM) has received considerable attention by fluid dynamic researchers. The LBM is based on the lattice Boltzmann equation with Bhatnagar-Groos-Krook (BGK) collision approximation. In this paper the incompressible laminar flow in a 2D micro semi-circular cavity with the lid, driven is simulated by LBM. It should be mentioned that the flow in semi-circular cavity in macro scale has also been simulated and the obtained results were found to be in good agreement with those given from the finite volume method. In the present work, the computational results showed that the slip could have influence on the centre of vortex and actually moved it in horizontal and vertical directions in semi-circular cavity. Computing the friction coefficient on the lid driven circular cavity presented that the friction coefficient was increased as Kn was increased. The slip also had a decreasing effect on the maximum velocity in the cavity.

Keywords: Micro Semi-circular Cavity, Knudsen Number, Slip Boundary Condition, Lattice Boltzmann Method

Reference: M. Alafzadeh and Sh. Talebi, (2012) 'Numerical Simulation of Laminar flow in Micro Semi-circular Cavity by LBM', Int. J. Advanced Design and Manufacturing Technology, Vol. 5/ No. 2, pp. 71-78

Biographical notes: M. Alafzadeh is Ph.D. student in Department of Mechanical Engineering, Isfahan University of Technology, Isfahan, Iran. **Sh. Talebi** is Associate Professor in Department of Mechanical Engineering, Yazd University, Yazd, Iran.

1 INTRODUCTION

According to the numerical progress in the world, micro and Nano scale particles have created an increasing interest in engineering and scientific researchers. These particles are very important and they have different environmental and industrial applications. When the characteristic size decreases down and it becomes a value comparable to the mean free path of the molecules, in this situation the fluid flow is not continuum and the Navier-stokes equations with no-slip boundary condition cannot be applied. Because the rarefaction effect becomes important and slip on the solid surface could affect the drag force. There are some methods to analyze the flow over micro particles. The numerical method which is not based on continuity of flow should be used to solve the flow over these particles. The Lattice Boltzmann Method (LBM) is an effective computational tool for the simulation of complex flows with no flow continuity to be imposed upon. Actually the LBM is a simplified solver of the Boltzmann equation on a discrete lattice. In [1] different boundary conditions and fundamental principle in the lattice Boltzmann method have been explained completely. In [2] the flow in a square cavity has been simulated in micro scale, in this paper after validating the method in macro scale, the effect of Knudsen number has been shown on friction coefficient. It was shown that Knudsen number could decrease friction coefficient. In [3] the flow over a curved boundary has been investigated in macro scale numerically, in this paper different boundary conditions for curved boundary have been demonstrated and compared with each other. In [4] the viscous flow has been simulated in semi-circular cavity in macro scale for high Reynolds number, in fact the effect of Reynolds number has been shown on stream lines and growth of vortex in cavity. To complete the simulation of semi-circular cavity by LBM, In this paper the 2D incompressible flow in a micro semi-circular cavity is solved by the Lattice Boltzmann method. The analysis is based on the D_2Q_9 lattice and Bhatnagar-Groos-Krook (BGK) kinetic equation.

2 FLOW GOVERNING EQUATION

Although the lattice Boltzmann method has been derived from the lattice gas, He and Luo [5] showed that this method could be gained from Boltzmann equation, the fluid is considered as a number of particles which collide and stream on specific links. In this method, D2Q9 model has been used as a lattice to discrete the fluid (Fig. 1). Distribution function's changes are derived from Boltzmann equation. Boltzmann equation by BGK approximation has the form [1]:

$$f_k\left(\vec{x}_i+\vec{e}_k,t+1\right)-f_k\left(\vec{x}_i,t\right)=-\frac{1}{\tau}\left[f_k\left(\vec{x}_i,t\right)-f_k^{eq}\left(\vec{x}_i,\right.\right. \qquad (1)$$

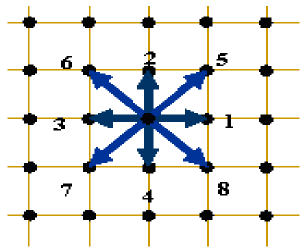

Fig. 1 D_2Q_9 Lattice

τ is the relaxation time, f^{eq} is the equilibrium distribution function that for D2Q9 model is expressed as [1]:

$$f_k^{eq}=\rho\omega_k\left[1+3\vec{e}_k.\vec{u}+\frac{9}{2}\left(\vec{e}_k.\vec{u}\right)^2-\frac{3}{2}\left|\vec{u}\right|^2\right]$$
$$\omega_0=4/9 \qquad \omega_2=\omega_4=\omega_6=\omega_8=1/36 \qquad (2)$$
$$\omega_1=\omega_3=\omega_5=\omega_7=1/9$$

In this equation [1]:

$$\vec{e}_0=(0,0)\quad i=1,2,3,4\quad \vec{e}_i=c\left(\cos\frac{(i-1)\pi}{2},\sin\frac{(i-1)\pi}{2}\right)$$
$$i=5,6,7,8\quad \vec{e}_i=c\sqrt{2}\left(\cos(\frac{(i-5)\pi}{2}+\frac{\pi}{4}),\sin(\frac{(i-5)\pi}{2}+\frac{\pi}{4})\right) \qquad (3)$$

f_k is the distribution function which is shown in Fig. 2. The flow quantities can be evaluated as:

$$\rho=\sum_{i=0}^{8}f_i \qquad \vec{u}=\frac{1}{\rho}\sum_{i=0}^{8}f_i\vec{e}_i \qquad \nu=\frac{1}{3}\left(\tau-\frac{1}{2}\right) \qquad (4)$$

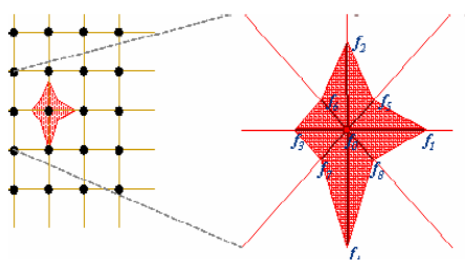

Fig. 2 Distribution function

When the characteristic length of flow is comparable to the mean free path of fluid, the rarefaction effect must be considered. So, the no slip boundary condition can not be applied for wall boundaries. The Knudsen number (Kn) is used to identify the rarefied phenomenon. The Kn number is the ratio of the mean free path (λ) to the characteristic length (L) of the flow. In this case the continuum assumption fails and the Navier-stokes equations with no slip boundary conditions cannot be applied. In such situations intermolecular collisions play a prominent role and the flow properties will be affected by the Knudsen number. It should be noticed that in micro scale, the relaxation time should be computed by Knudsen number instead of viscosity as follows [1]:

$$\tau = Kn.N + 0.5 \tag{5}$$

In this problem the characteristic length is the length of the driven lid (D).

For a low value of Mach number the Kn and Re numbers are inversely proportional. Thus Re number can not change arbitrarily for a given Ma where the slip flow regime stands. For general flow the relation of Kn number, Re number and Ma number are according to:

$$Kn = \sqrt{\frac{\pi\gamma}{2}}\frac{Ma}{Re} \tag{6}$$

For incompressible flow, Ma number must be less than 0.2, thus in the selection of proper values for Kn and Re it should be noted that Ma is less than 0.2. One of the comparable factors in this paper is the drag coefficient so proper evaluation of the fluid force is essential.

A body-fitted coordinate system together with grid stretching was used such that a large number of nodes can be placed near the body. The deviator stress for incompressible flow is calculated as:

$$\tau_{ij} = \rho\nu\left(\partial_i u_j + \partial_j u_i\right) \tag{7}$$

and can be evaluated using the non-equilibrium part of the distribution function as [6]:

$$\tau_{ij} = \left(1 - \frac{1}{2\tau}\right)\sum_{\alpha} f_{\alpha}^{(neq)}\left(x,t\right)\left(e_{\alpha,i}e_{\alpha,j} - \frac{1}{D}e_{\alpha}\cdot e_{\alpha}\delta_{ij}\right) \tag{8}$$

In this equation f^{neq} is the non-equilibrium part and is equal to f_{α}-f^{eq}. Instead of stress integration method, Ladd [6] used the momentum exchange method to compute the fluid force. The total force acting on a solid body is obtained as:

$$F = \sum_{all x_S}\sum_{\alpha\neq 0} e_{\alpha}\left[\tilde{f}_{\alpha}\left(x_S,t\right) + \tilde{f}_{\bar{\alpha}}\left(x_f,t\right)\right]\times\left[1-\omega(x_f)\right]\times\frac{\delta\forall}{\delta t}$$

$$\omega\left(x_f\right) = \begin{cases} 0 & \text{in solid body} \\ 1 & \text{out of body} \end{cases} \tag{9}$$

where $\omega\left(x_f = x_s + e_{\bar{\alpha}}\right)$ is an indicator, which is zero at x_f (fluid node) and one at x_s (solid node) (Fig. 3) $\tilde{f}_{\alpha}$ is a distribution function in post collision.

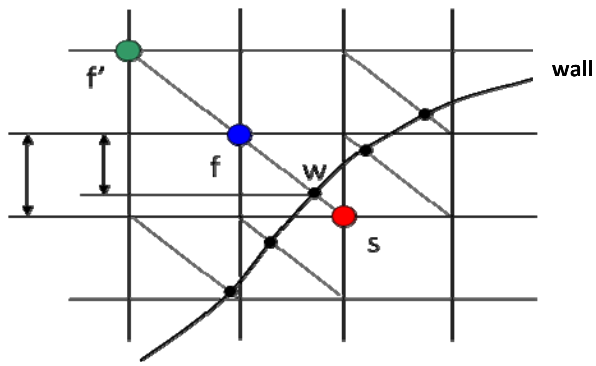

Fig. 3 Solid grid & fluid grid near the surface

It should be mentioned that in this paper the Bouzidi and Yu method was used for curved surface. Implementing the Bouzidi method, the resultant can be defined as equation group 10 [7]:

$$\tilde{f}_{-i}\left(r_S,t\right) = 2q.\tilde{f}_i(r_S,t) + (1-2q)\tilde{f}_i(r'_F,t) \qquad q <$$

$$\tilde{f}_{-i}(r_S,t) = \frac{1}{2q}\tilde{f}_i(r_F,t) + \frac{(2q-1)}{2q}\tilde{f}_{-i}(r_F,t) \qquad q \geq \frac{1}{2} \tag{10}$$

Implementing the the Yu method, the resultant can be defined as equation 11 [8]:

$$\tilde{f}_{-i}(r_S,t) = \frac{1}{1+q}\left[q.\tilde{f}_i(r_F,t) + (1-q).\tilde{f}_i(r'_F,t) + q.\tilde{f}_{-i}(r_F,t)\right] \tag{11}$$

In the two previous equation groups q was identified as:

$$q = \frac{\left|r_F - r_W\right|}{\left|r_F - r_S\right|} = \frac{\left|r_F - r_W\right|}{\Delta x} \qquad 0 < q \leq 1 \tag{12}$$

The Bouzidi method could be used in micro scale. In fact after computing the slip velocity on the surface by interpolation, it could be used as u_w in the Bouzidi method. The DMBC method which was used for slip boundary condition is defined as [9]:

$$
\begin{aligned}
f_2 &= \sigma f_2^{eq} + (1-\sigma)\tilde{f}_8 \\
f_3 &= \sigma f_3^{eq} + (1-\sigma)\tilde{f}_7 \\
f_4 &= \sigma f_4^{eq} + (1-\sigma)\tilde{f}_6
\end{aligned}
\qquad (13)
$$

where $\tilde{f}$ is a distribution function in post collision.

3 NUMERICAL SIMULATION

3.1. Macro scale

The flow geometry has been shown in Fig. 4. Computational grids were considered 75×150. These solution domain and computational grids were the best grids which were derived after using different computational grids and solution domains. The upper boundary was considered as a driven lid and the other walls were supposed as a curved surface. In macro scale the lid driven upper wall was treated by extrapolation and the Yu method was used to show the no slip on the curved surface The momentum exchange has been used to compute the force exerted upon the solid surface. It should be mentioned that the Reynolds number is defined by diameter of the cavity and velocity of the upper wall (lid driven).

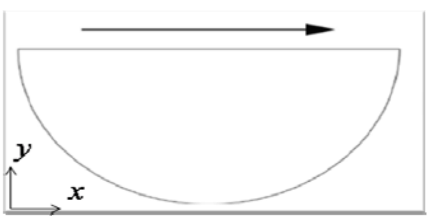

Fig. 4 Flow in semi-circular cavity

At first this problem was investigated in macro scale to clarify the computational code in LBM in comparison with other numerical result. In this paper finite volume is a numerical method used as a reference. It should be mentioned that the number of grids in the two methods are the same to have a better comparison.

The stream line for Re=100 was depicted in Fig. 5 in comparison with numerical result (finite volume) as it was shown, two stream lines are the same so the computational result is accurate enough. The resultant center obtained from the LBM method is x/D=0.6081 and the finite volume proves this center to be x/D=0.6167.

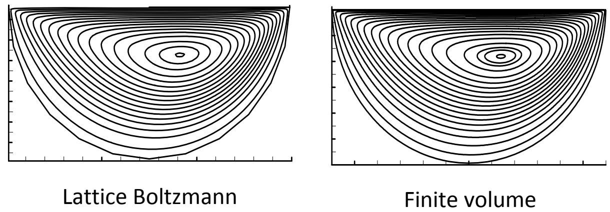

Fig. 5 Stream lines in semi-circular cavity

It is clear that the LBM result is very close to the finite volume result. In Fig. 6 the horizontal component velocity along a vertical line in the centre of cavity was shown. This figure shows that the computational result is accurate. This result could show that the fluid turns in the cavity. It is clear that because of vortex in cavity, the sign of horizontal component of velocity has changed. The velocity on the driven lid is equal to 1 because there is no slip on the upper wall.

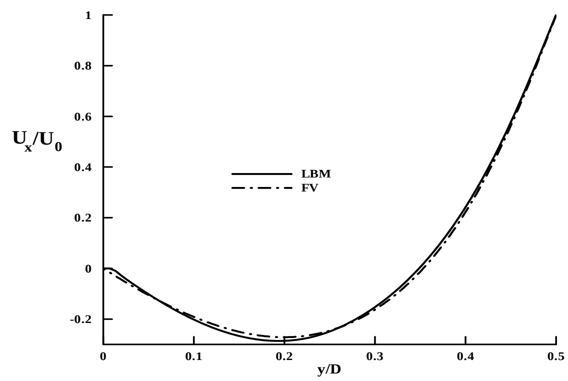

Fig. 6 Horizontal velocity component distribution along Y axis in center

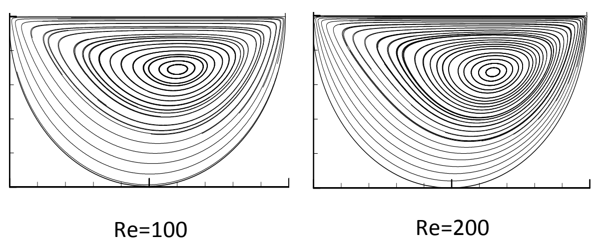

Fig. 7 Effect of Re number on stream lines

To show the accuracy of LBM, the effect of Re number on the center of vortex is shown in Fig. 7. As it was expected when the Re number increases the center of vortex moves to the left.

3.2. Micro scale

In this part the flow in the semi-circular cavity was simulated in micro scale with the same dimension. In micro scale, all the boundaries were treated by DMBC (Discrete Maxwell Boundary Condition) which is a straight forward discretization of Maxwell's diffuse reflection boundary condition in kinetic theory. According to equation 6 for a small Mach number, the Re number was considered 3.46 in this problem.
Table 1 and Fig. 7 show the effect of Kn number on the friction coefficient and the horizontal component velocity respectively. It should be mentioned that this coefficient is defined as the force exerted upon the moving wall (equation 14):

$$C_f = \tau\sqrt{2/3}\,/\,2P_0U_0 \tag{14}$$

P_0 is pressure and τ is tangential stress as it was depicted, when Kn number increases the friction coefficient increases too. Because the slip on the driven lid will increase when the Kn number increases so more force is needed to resist the lid movement.

Table 1 The effect of Knudsen number on friction coefficient in semi-circular cavity

Knudsen number	Kn= 0.01	Kn= 0.03	Kn= 0.05	Kn= 0.07	Kn= 0.09
Friction coefficient	0.068	0.158	0.194	0.221	0.222

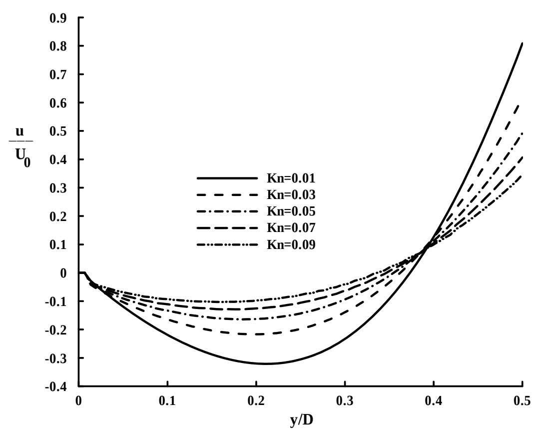

Fig. 8 The effect of Knudsen number on horizontal velocity component in the center

In Fig. 8 the variation of horizontal velocity component at x/D=0.5 is shown. When Knudsen number increases, the maximum velocity will decrease. High Kn number means the rarefaction effect is high, so the upper fluid layer has less effect on the lower layer. While the gradient of velocity on the upper wall decreases as Kn number increases, the friction coefficient decreases. higher Kn number results in lower density and decreased friction coefficient.

4 CONCLUSION

In this paper the 2D incompressible flow in semi-circular cavity is simulated in macro and micro scale. As it was shown Re number could move the centre of vortex in macro scale. In micro scale, when Kn number increases, the maximum velocity and friction coefficient decrease. The results show the accuracy of lattice Boltzmann method to simulate complex geometries in macro and micro scale.

REFERENCES

[1] Succi, S., The lattice Boltzmann Equation for Fluid Dynamics and Beyond, Oxford University Press, 2001.
[2] Naris, S. and Valougeorgis, D., "The Driven Cavity Flow over the Whole Range of the Knudsen Number", Physics of Fluids, Vol. 17, 2005, pp. 33-39.
[3] Mei, R., Luo, L. S. and Shyy, W., "An Accurate Curved Boundary Treatment in the Lattice Boltzmann Method", Journal of Computational Physics, Vol. 155, 1999, pp. 307-330.
[4] Glowinski, R., Guidoboni, G. and Pan, T. W, "Wall Driven Incompressible Viscous Flow in a Two-dimensional Semi-circular Cavity", Journal of Computational Physics, Vol. 216, 2006, pp. 76-91.
[5] He, X. and Luo, L. S., "Theory of the Lattice Boltzmann Method: from the Boltzmann Equation to the Lattice Boltzmann Equation", Physical Review E, Vol. 56, No. 6, 1997, pp. 6811-6817.
[6] Filippova, O. and Hanel, D., "Grid Refinement for Lattice-BGK Models", Journal of Computational Physics, Vol. 147, pp. 219-228, 1998.
[7] Bouzidi, M., Firdaouss, M. and Lallemand, P., "Momentum Transfer of Boltzmann-lattice Fluid with Boundaries", Physics of Fluids, Vol. 13, 2001, pp. 3452-3459.
[8] Yu, D., Mei R. and Shyy, W., "A Multi-block Lattice Boltzmann Method for Viscouse Fluid Flows", International Journal of Numer. Methods Fluids, Vol. 39, 2002, pp. 99-120.
[9] Guo, Zh., Shi, B., Zhao, T. S. and Zheng, Ch., "Discrete Effects on Boundary Conditions for the Lattice Boltzmann Equation in Simulating Microscale Gas Flows", Physics Review E, Vol. 76, 2007, pp. 35-40.

SIMBOLS

Lattice velocity	*c*

Spead of sound	c_s
Relaxation time	τ
distribution functions	f_k
equilibrium distribution functions	f^{eq}
distribution function in post collision	$\tilde{f}$
non equilibrium distribution functions	f^{neq}
Knudsen number	Kn
Reynolds number	Re
Density	ρ
Link	k
Node	i
Force	F
Discrete velocity	$\vec{e}_i$
Mach number	Ma
Stress tension	τ_{ij}
Friction coefficient	C_f