UDC 532.5

Stability investigation of the two-dimensional nine-vectors model of the lattice Boltzmann method for fluid flows in a square cavity

G. Bulanchuk, O. Bulanchuk, A. Ostapenko
Pryazovsky State Technical University, Ukraine

In this paper we consider the stability of the two-dimensional nine-vectors model of the lattice Boltzmann method which used to model fluid flows in a square lid-driven cavity. Obtained numerical solutions were compared with the results of the numerical experiments by the finite element method. We investigate the influence of Reynolds and Mach numbers on method`s stability. Shown the dependence between the kinematic viscosity of the modeling liquid and cell`s size. Have been shown the advantages and disadvantages of this computational method.

Key words: *Navier-Stokes equation, kinetic theory, particle, Reynolds number, Mach number.*

В роботі розглядається застосування двовимірної дев'ятишвидкісної моделі методу граткових рівнянь Больцмана до моделювання течії в'язкої рідини у квадратній каверні з рухомою верхньою стінкою. Отримані результати порівнювались із результатами чисельних експериментів методом скінчених елементів. Досліджується вплив числа Рейнольдса та числа Маха на стійкість методу. Показана залежність в'язкості рідини, що моделюється, з необхідним розміром комірки розрахункової сітки. Розглянуті переваги та недоліки даного підходу.

Ключові слова: *рівняння Навьє-Стокса, кінетичная теорія частка, число Рейнольдса, число Маха.*

В работе исследуется устойчивость двумерной девятискоростной модели метода решеточных уравнений Больцмана на примере моделирования течения вязкой жидкости в квадратной каверне с движущейся верхней стенкой. Полученные численные решения сопоставляются с результатами численных экспериментов методом конечных элементов. Исследуется влияние числа Рейнольдса и числа Маха на устойчивость метода. Показана зависимость между вязкостью моделируемой жидкости и необходимым размером ячейки расчетной сетки. Рассмотрены преимущества и недостатки данного подхода.

Ключевые слова: *уравнение Навье-Стокса, кинетическая теория, частица, число Рейнольдса, число Маха.*

1. Introduction

There are two classes of methods to simulate fluid flows: mesh-based and mash-free ones. Among mesh-based methods widely used the finite difference method (FDM) [1], the finite element method (FEM) [2] and the finite volume method (FVM) [3]. The discrete vortex method [4], the diffusion velocity method [5] and the smoothed-particle hydrodynamics method (SPH) [6] are the examples of mesh-free computational methods. Even so these methods have been proven their efficiency in many problems of hydrodynamics there are still many difficulties related with the numerical solution of the Navier – Stokes equations with high-Reynolds number [7,8].

Recently new computational method that combines the advantages of Euler (mesh-based methods) and Lagrangian (mesh-free methods) representation of fluid have been proposed [9]. In this work we shall discuss hybrid method based on the kinetic theory of gases – the lattice Boltzmann method (LBM) [10], which uses the discrete

© Bulanchuk G., Bulanchuk O., Ostapenko A., 2015

Boltzmann equation to simulate flows of viscous liquid [11]. The lattice Boltzmann method appeared in 1990 and it is keep rapidly growing up. Have been made some commercial (PowerFlow, XFlow) and free-available (Palabos) software for computational fluid dynamics with the LBM models.

The aim of this work is to investigate the stability of numerical solutions for the laminar flows in a square lid-driven cavity, that were get using the two-dimensional nine-vectors model of the lattice Boltzmann method [11]. It should be noted, that some solutions for lid-driven cavity by LBM were obtained in [12, 13, 14]. But the stability problem in these works wasn't investigate. Received by the LBM results were compared with the numerical solutions that were get by the finite element method. The photo of the flow in a cavity, which has got during the experiment shown in fig. 1.

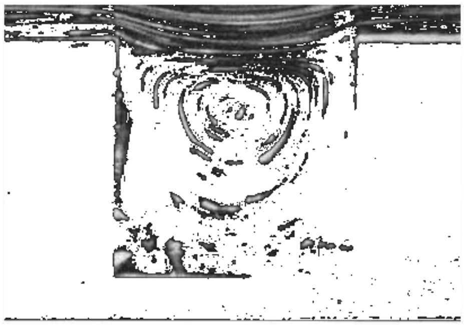


Figure 1 –The creeping flow in a square cavity. Re=0.01[15]

2. The Lattice Boltzmann Method: D2Q9 BGK model

According to the LBM, the computational domain divides into rectangular cells to create the lattice. The fluid flows dynamics are treated as pseudo particles dynamics, which are in the cells of the lattice [16]. Such pseudo particles can move between cells only by determined directions defined by the lattice model [17] (fig. 2). We used the model specification like DpQn, where $p = \{1,2,3\}$ denotes the dimension of the lattice and $n \in N$ denotes the number of vectors in the lattice.

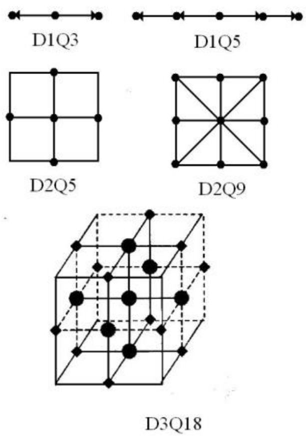


Figure 2 – Lattice models

The dynamics of pseudo particles describes by the kinetic theory of gasses using the discrete single particle densities distribution function $f_k\left(\vec{r},t\right)$ [18]. The densities distribution function f_k determines density ρ and velocity vector $\vec{u}$ of the liquid for each cell according to equations [19]:

$$\rho\left(\vec{r},t\right)=\sum_{k=1}^{n} f_k\left(\vec{r},t\right); \qquad \vec{u}\left(\vec{r},t\right)=\frac{1}{\rho}\sum_{k=1}^{n}\vec{e}_k f_k\left(\vec{r},t\right) \tag{3.1}$$

where ρ – liquid density;

$\vec{r}=\left(x,y\right)$ – coordinates;

t – time;

$\vec{u}$ – velocity vector;

$\vec{e}_k$ – possible directions vector for pseudo particles movement;

We shall use the Bhatnagar – Gross – Krook (BGK) collision operator [18] to approximate particles collision (perfectly elastic collision). It is a linear relaxation to local Maxwell equilibrium function:

$$I_k=\frac{f_k^{eq}\left(\rho,\vec{u}\right)-f_k\left(\vec{r},t\right)}{\tau} \tag{3.2}$$

where f_k^{eq} – equilibrium Maxwell-Boltzmann distribution function [19];

τ – dimensionless relaxation parameter ($\tau>0.5$)[17], connected with the kinematic viscosity of the liquid [19].

$$\nu=\frac{(2\tau-1)}{6}c\Delta x \tag{3.3}$$

c – the base speed of the cell [19], calculated according to the equation:

$$c=\frac{d}{\Delta t} \tag{3.4}$$

where d – space step

Δt – time step.

The most common 2D lattice model is D2Q9. Fig. 3 shows the possible directions for particles movement $\vec{e}_k$. The direction $\vec{e}_0$ is the state of the rest.

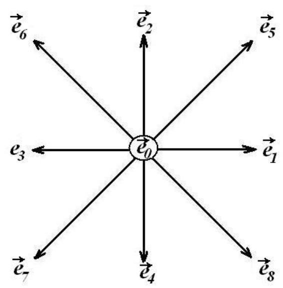


Figure 3 – The possible particles movement directions for the D2Q9 lattice model

The system of the discrete kinetic equations describing the dynamics of the particles has the form [17]:

$$f_k\left(\vec{r}+\vec{e}_k\Delta x, t+\Delta t\right)=f_k\left(\vec{r},t\right)-\frac{1}{\tau}\left[f_k\left(\vec{r},t\right)-f_k^{eq}\left(\rho,\vec{u}\right)\right] \tag{3.5}$$

To model the isothermal flows with LBM we shall use the expansion of the equilibrium Maxwell distribution function by the powers of the velocity vector [17]:

$$f_k^{eq}=w_k\rho\left(1+\frac{3}{c^2}\left(\vec{e}_k,\vec{u}\right)+\frac{9}{2c^4}\left(\vec{e}_k,\vec{u}\right)^2-\frac{3}{2c^2}\vec{u}^2\right) \tag{3.6}$$

where w_k – weights;

For the D2Q9 lattice model the weights are: $w_0=\frac{4}{9}; w_{1-4}=\frac{1}{9}; w_{5-8}=\frac{1}{36}$ [19].

In the LBM there is a constant depending on the lattice – speed of sound [19]. In the D2Q9 model it can be calculated using the following equation:

$$c_s=\frac{c}{\sqrt{3}} \tag{3.7}$$

The liquid pressure connects with the speed of sound by the next equation:

$$p=c_s^2\rho \tag{3.8}$$

According to [13,14,18] we shall set $c=1$. So $c_s=\frac{1}{\sqrt{3}}$.

Software implementation of the statistical approach that uses the kinetic theory of gases to describe fluid dynamics is quite simple and intuitive. But the disadvantage of such approach is instability. It can appear because of:

1. τ – relaxation time. To avoid negative influence of a relaxation time parameter in this work we shall set $\tau=1$ [17]. So according to eq. (3.3) the kinematic viscosity will uniquely determines the size of the cells of the computational grid.
2. c_s – speed of sound. Based on [18] the LBM is stable when $c_s<\sqrt{1-U_{\max}^2}$. In this work this condition makes the velocity bound $U_{\max}<0.81$.
3. $M=\frac{\left|\vec{u}\right|}{c_s}$ – Mach number. According to [19] there is a limitation: $M<<1$.

3. The flows in a cavity

Let us consider a square cavity with L = 1 m which has three stationary sides and one moving side (fig. 4).

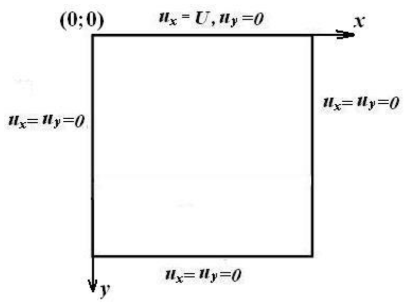


Figure 4 – Cavity configuration

We used various values of the moving wall $U = 0.1, 0.3, 0.7$ so we got various Reynolds numbers $\mathrm{Re} = 120, 360, 840$ and various Mach numbers $M = 0.1, 0.3, 0.7$. This problem was solved not only with the LBM, but with the finite element method (FEM) in the Comsol Multiphysics 4.2 package too. The solutions of the same tasks were compared. The calculations were made using the uniform 200x200 grid with the LBM and using "normal" grid with the Comsol package (1504 cells).

The velocity u_x component contours and velocity magnitude contours u for steady flow in a cavity shown in fig. 5. The parameters of the flow are $U = 0.1, \mathrm{Re} = 120, M = 0.1$

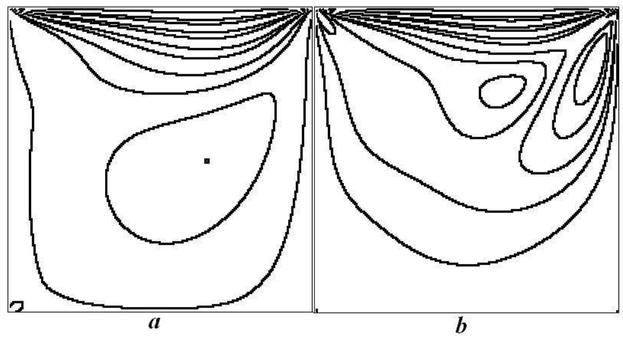


Figure 5 – Velocity contours that were get with the LBM
u_x velocity component (a), velocity magnitude u (b). Re=120.

Received numerical results were compared with the same ones got in the Comsol package. Fig. 6 illustrates velocity graphics in different sections of the cavity $x = 0.25, 0.75, 0.75$. Here and then solid lines correspond to the solutions got with the lattice Boltzmann method (LBM) and points correspond to the solutions got with the finite element method (FEM). The vertical axis contains velocity values and the horizontal axis contains y coordinates of the cavity.

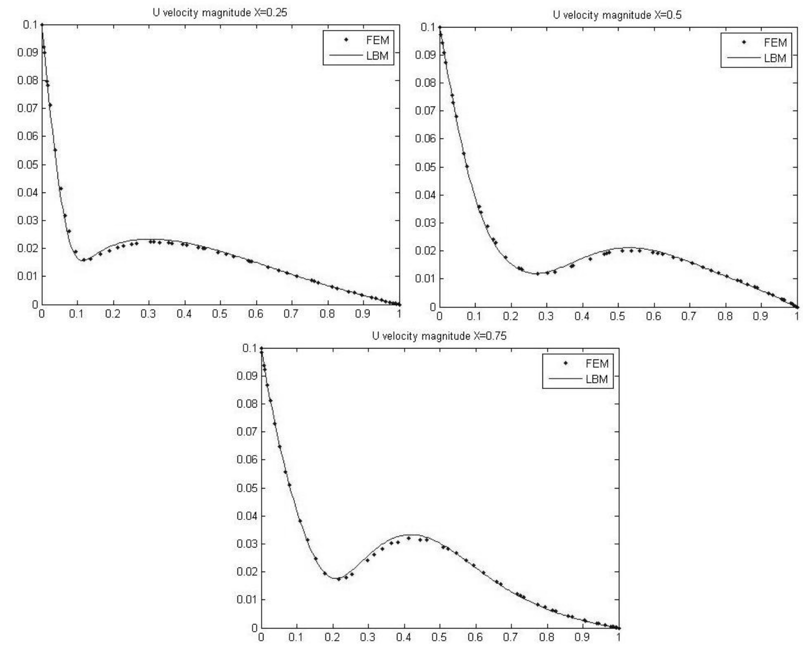


Figure 6 – The velocity magnitude graphs in the x=0.25,0.5,0.75 sections of the cavity. Re = 120

The similar results had got for the u_x velocity component (fig. 7).

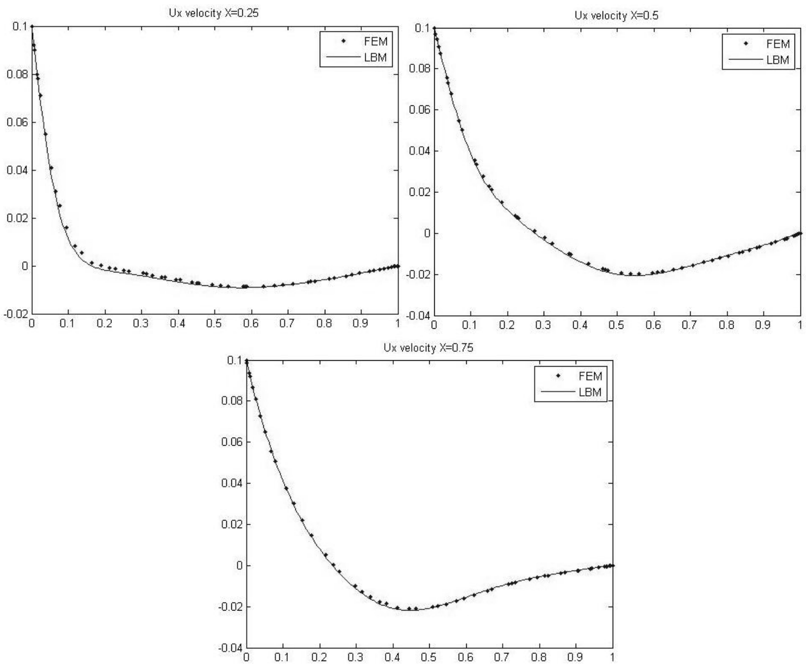
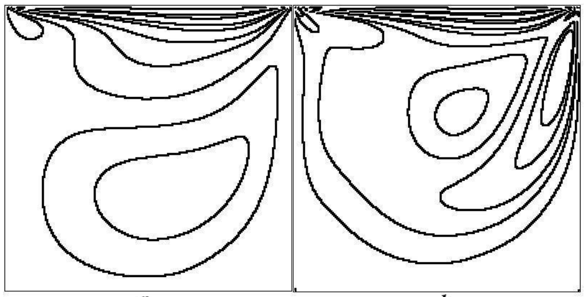


Figure 7 – The u_x velocity graphs in the x=0.25,0.5,0.75 sections of the cavity. Re = 120

The steady flow in a cavity with $U = 0.3, \mathrm{Re} = 360, M = 0.3$ illustrated in fig. 8.



a *b*

Figure 8 – Velocity contours that were get with the LBM u_x velocity component (a), velocity magnitude u (b). Re=360.

The velocity magnitude graphics in different sections of the cavity $x = 0.25, 0.75, 0.75$ shown in fig. 9. The similar u_x component velocity graphs shown in fig. 10.

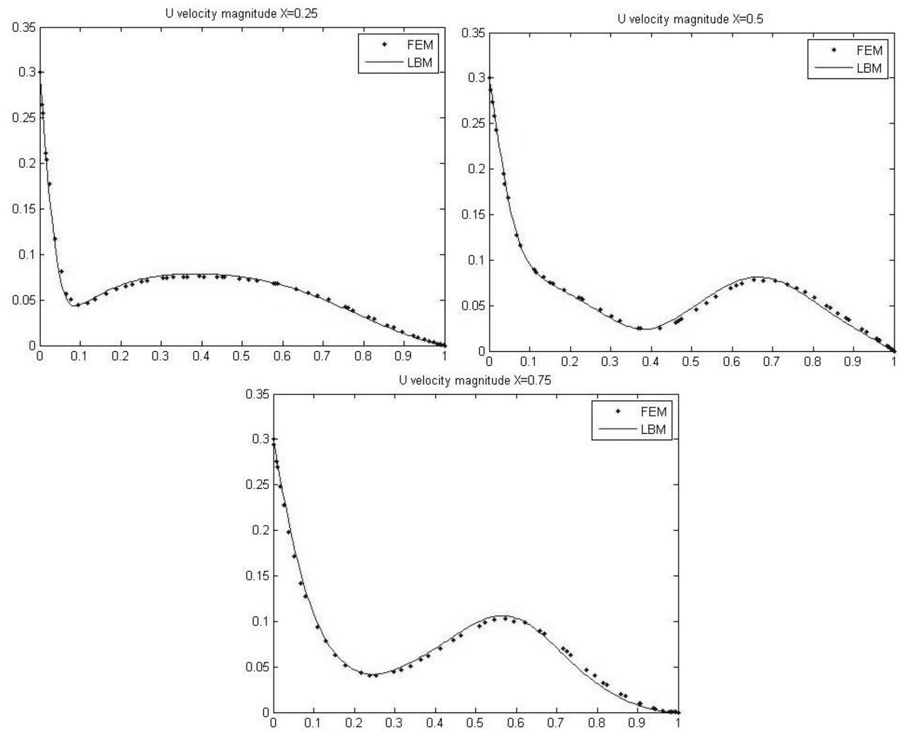


Figure 9 – The velocity magnitude graphs in the x=0.25,0.5,0.75 sections of the cavity. Re = 360

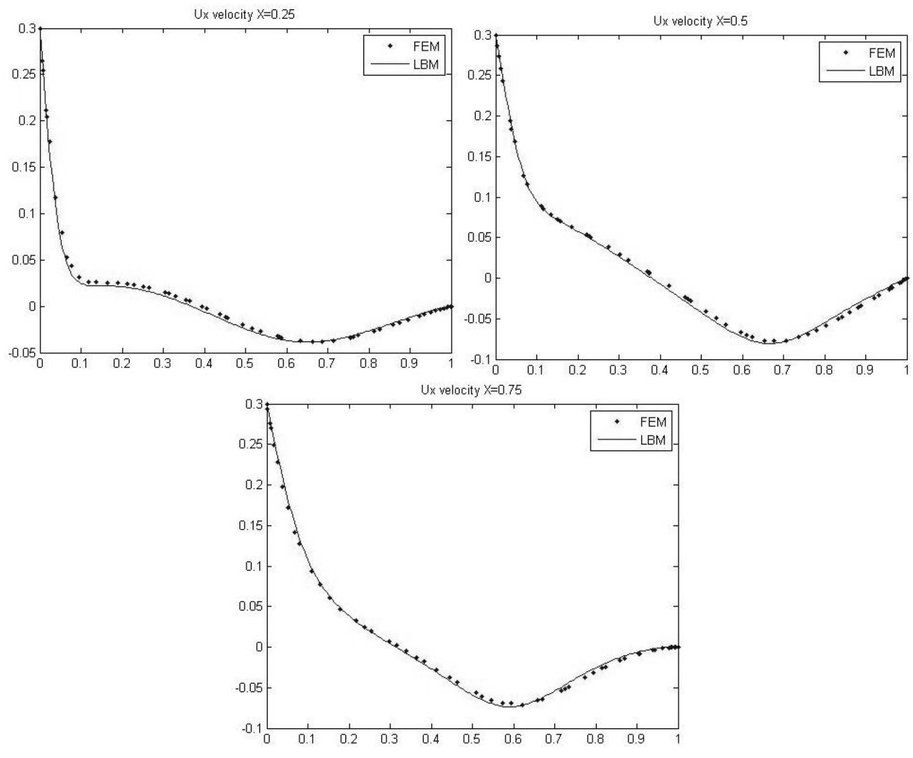
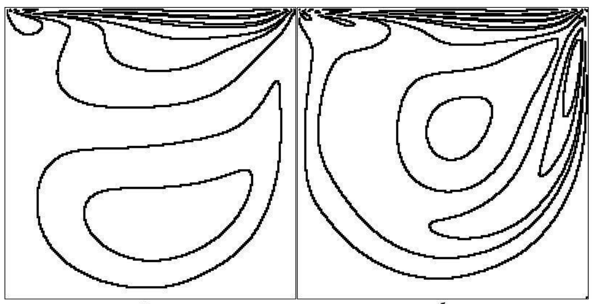


Figure 10 – The u_x velocity graphs in the x=0.25,0.5,0.75 sections of the cavity. Re = 360

You can see the steady flow (velocity contours) in a cavity with $U = 0.7, \mathrm{Re} = 840, M = 0.7$ in fig. 11.



a ***b***

Figure 11 – Velocity contours that were get with the LBM
u_x velocity component (a), velocity magnitude (b). Re = 840

Fig. 12, 13 shows the velocity magnitude u and the u_x velocity component graphs in different cavity sections.

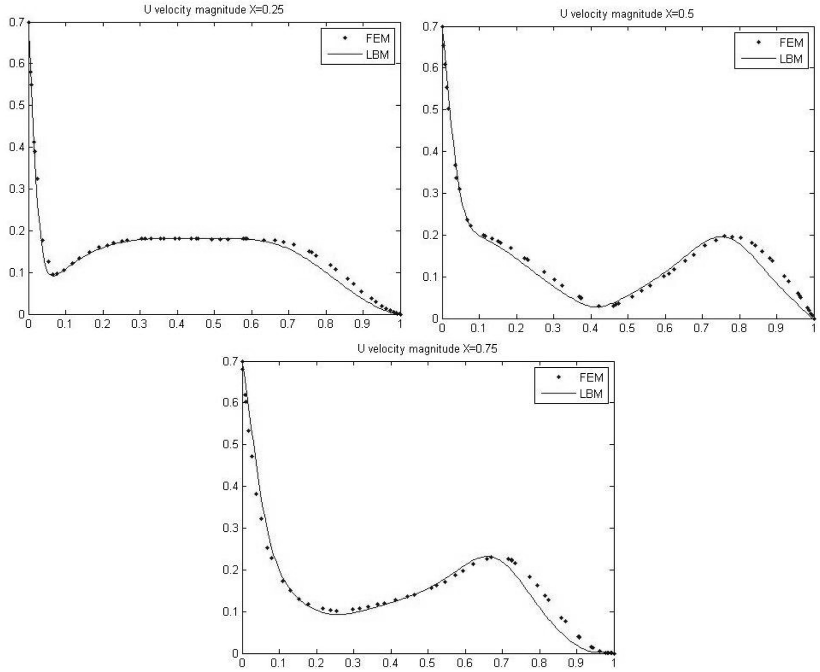


Figure 12 – The velocity magnitude graphs in the x=0.25,0.5,0.75 sections of the cavity. Re = 840

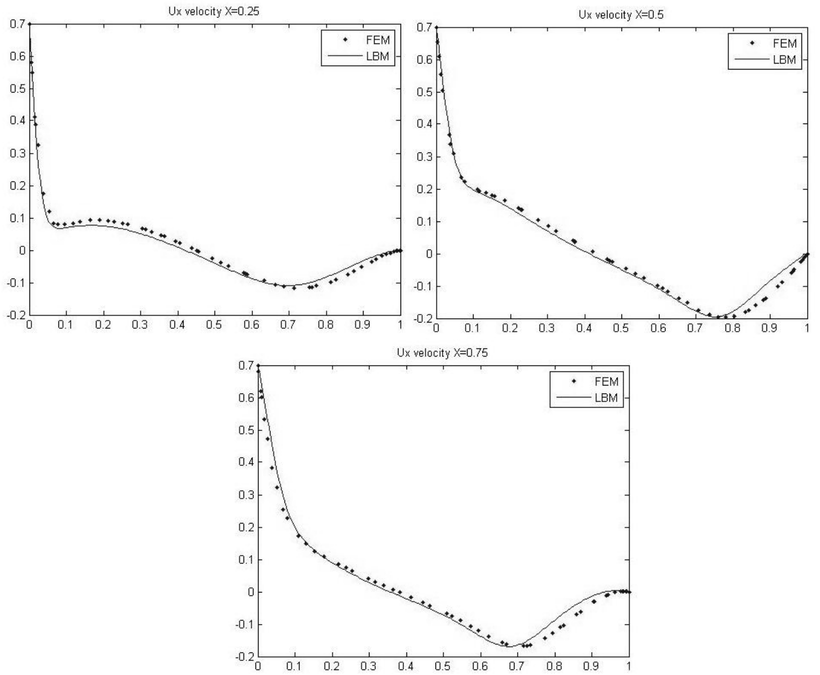
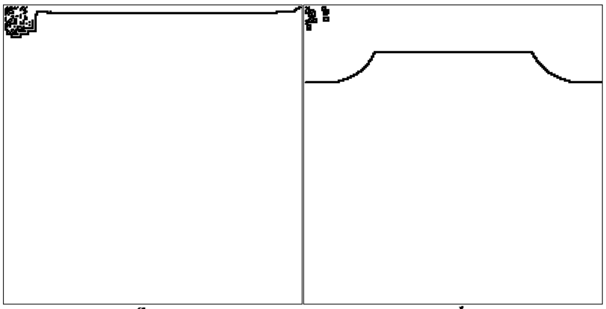


Figure 13 – The u_x velocity graphs in the x=0.25,0.5,0.75 sections of the cavity. Re = 840

As shown in fig. 6, 7, 9, 10, 12, 13 the graphs of numerical solutions gradually diverge with the Mach number increasing (from 0.1 to 0.7). It is because to get a stable solution with the LBM the Mach number should be much less than one. In practice, the recommended value of the Mach number is less than 0.15 ($M < 0.15$) [19].

Another problem is the limitation of the maximum possible speed of fluid (see the previous section). The numerical solutions that were get with the LBM using 200x200 grid are completely at odds when $U \geq 0.72$ (fig. 14).

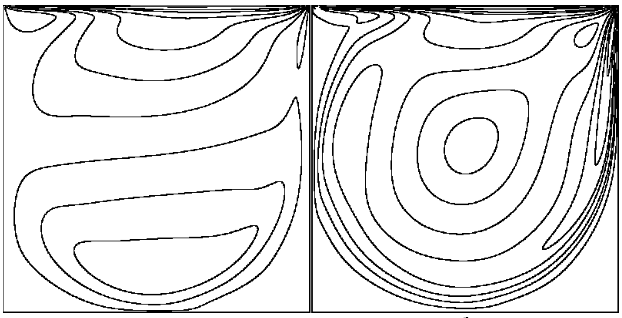


a *b*

Figure 14 – Velocity contours that were get with the LBM
u_x velocity component (a), velocity magnitude (b). Re = 864

The maximum Reynolds number at which the solution remains stable at the 200x200 grid is $\mathrm{Re} = 840$.

Let's consider the numerical results got with the LBM with 600x600 grid. The velocity magnitude u contours and u_x velocity component contours for steady flow in a cavity shown in fig. 15 for $U = 0.6, \mathrm{Re} = 2160, M = 0.6$.



a *b*

Figure 15 – Velocity contours which had got with LBM
u_x velocity component (a), velocity magnitude (b). Re = 2160

Fig. 16 and 17 shows the velocity magnitude u graphs and u_x velocity component graphs in different sections of the cavity.

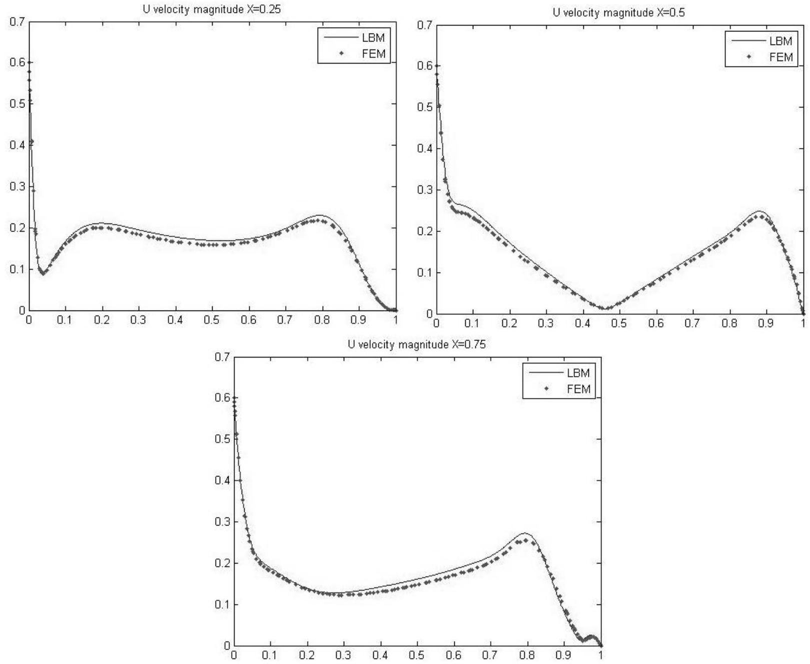


Figure 16 – The velocity magnitude graphs in the x=0.25,0.5,0.75 sections of the cavity. Re = 2160

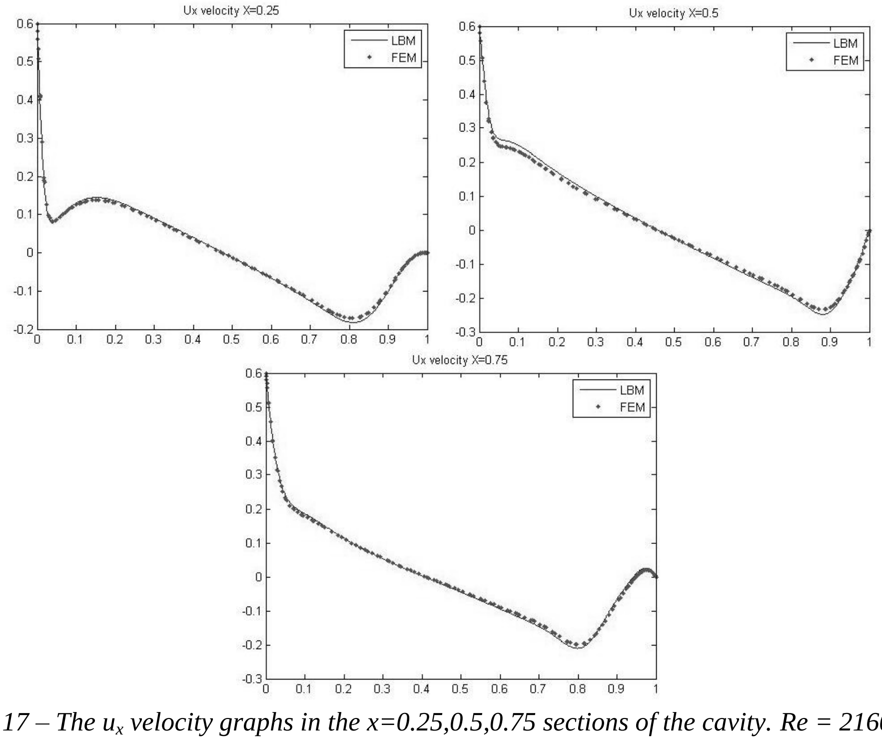


Figure 17 – The u_x velocity graphs in the x=0.25,0.5,0.75 sections of the cavity. Re = 2160

So the maximum Reynolds number at which the solution remains stable depends on the size of the computational grid (fig. 18).

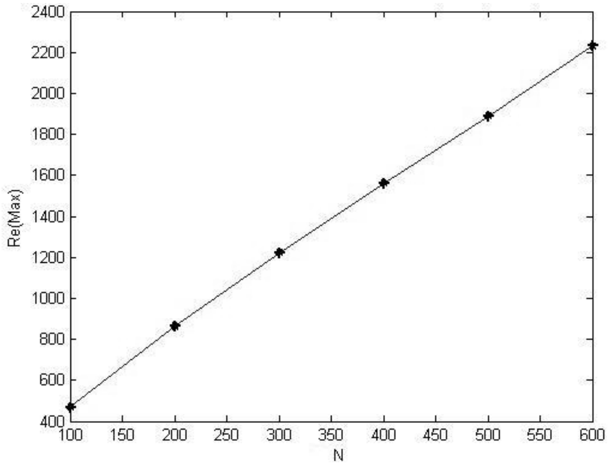


Figure 18 – The graph of the dependence between the critical Reynolds number and the computational grid size

Unfortunately, using 600x600 grid we can`t model flows with Reynolds number $\mathrm{Re} > 2160$, but it is not the LBM limit. To model flows with bigger Reynolds number we should do mesh finer. So according to the introduced assumptions the viscosity will change (eq. 3.7). Obviously, that such approach would lead to increasing the calculation time. But this disadvantage can be neutralized by the parallel computing on powerful multicore processors or by using the cloud computing services.

4. Conclusion

Based on the numerical results, that were get with the lattice Boltzmann method and the finite element method we can tell that the LBM provides high accuracy solutions when $M < 0.3$. The results with $0.3 < M < 0.7$ showed some deviations that increases with Mach number increasing, but no more than 10%.

Was defined the critical Reynolds number at which the numerical solutions remains stable. This number depends on the size of the computational grid. Therefore, flows modeling with high Reynolds numbers take more time.

Although the lattice Boltzmann method can become instable the method has great potential and has a lot of significant advantages. The next step in our research is to modify the algorithm to avoid instability in solutions with high Reynolds number and model fluid flows with more complex geometry.

REFERENCES

1. G. D. Smith Numerical Solution of Partial Differential Equations: Finite Difference Methods. – Oxford: Univercity Press, 1986. – 350 p.
2. G. Strang An Analysis of The Finite Element Method. Prentice Hall. – Prentice Hall, 1973. – 400 p.

3. R. Eymard, T.R. Gallouet, R. Herbin The finite volume method Handbook of Numerical Analysis. – Paris, 2000. – 1020 p.
4. Белоцерковский С. М., Скобелев Б. Ю. Метод дискретных вихрей и турбулентность. – Новосибирск: ИТПМ, 1993. — 38 с.
5. Y. Ogami, T. Akamatsu Viscous flow simulation using the discrete vortex model - the diffusion velocity method // Computers & Fluids. — 1991. — Vol. 19, no. 3/4. — 433-441 p.
6. J.J. Monaghan An introduction to SPH// Computer Physics Communications. – 1988. – vol. 48. – pp. 88-96.
7. A. J. Chorin Numerical Solution of the Navier-Stokes Equations// Mathematics of Computation. – 1968. – Vol.22, No.104. – 745-762 p.
8. D. O. Martinez, W. H. Matthaeus, S. Chen, D.C. Montgomery Comparison of spectral method and lattice Boltzmann simulations of two-dimentional hydrodynamics// Physics of Fluids. – 1994. – Vol.6, No 3. – P. 1285-1298
9. Белоцерковский О.М. Численное моделирование в механике сплошных сред. – М.: Наука, 1984. – 520 с.
10. L. S. Luo Theory of the lattice Boltzmann method: lattice Boltzmann models for nonideal gases// Physical Rexiew. – 2000. – Vol.62, No 4. – P. 4292-4996
11. S. Succi The lattice Boltzmann equation: a new tool for computational fluid dynamics//Physica D. – 1991. – V.47 №. 1. – 219–230 p.
12. C. Rettinger Fluid Flow Simulation using the Lattice Boltzmann Method with multiple relaxation times. – Bachelor, 2013. – 38 p.
13. M. Mussa Numerical Simulation of Lid – Driven Cavity Flow Using the Lattice Boltzmann Method// Applied Mathematics. – 2008. – 236-240 p.
14. Г. В. Кривовичев О расчете течений вязкой жидкости методом решеточных уравнений Больцмана// Компьютерные исследования и моделирование. – 2013. – Т.5 №2. – 165-178 стр.
15. M. Van Dyke An Album of Fluid Motion. – California: The Parabolic Press, 1982. – 184 p
16. S. Succi The Lattice Boltzmann Equation for Fluid Dinamics and Beyond. – Oxford: Univercity Press, 2001. – 288 p
17. M. Sucop Lattice Boltzmann Modeling. An Introduction for Geoscientists and Engineers. – Miami, 2006. – 171 p
18. D. Wolf-Gladrow Lattice-Gas Cellular Automata and Lattice Boltzmann Models - An Introduction . – Bremerhaven: Alfred Wegener Institute for Polar and Marine, 2005. – 273 p
19. X. He Lattice Boltzmann Model for the Incompressible Navier – Stokes Equation//Journal of statistical physics. – 1997. – V.88. – 927–944 p.

Надійшла у першій редакції 12.05.2015, в останній - 29.11.2015.