LATTICE BOLTZMANN METHOD FOR TWO-DIMENSIONAL UNSTEADY INCOMPRESSIBLE FLOW

Juraj MUŽÍK^{1*}

¹ Department of Geotechnics, Faculty of Civil Engineering, University of Žilina, Univerzitná 8215/1, 010 26 Žilina, Slovakia.

* corresponding author: muzik@fstav.uniza.sk.

Abstract

A Lattice Boltzmann method is used to analyse incompressible fluid flow in a two-dimensional cavity and flow in the channel past cylindrical obstacle. The method solves the Boltzmann's transport equation using simple computational grid - lattice. With the proper choice of the collision operator, the Boltzmann's equation can be converted into incompressible Navier-Stokes equation. Lid-driven cavity benchmark case for various Reynolds numbers and flow past cylinder is presented in the article. The method produces stable solutions with results comparable to those in literature and is very easy to implement.

Keywords:

Boltzmann's transport equation;
Lattice Boltzmann method;
Incompressible flows;
Navier-Stokes equations.

1. Introduction

The Lattice Boltzmann method shows the ability as a promising tool for modelling the Navier-Stokes equations and simulating fluid flow problems. LBM is based on mesoscopic statistical behaviour of the liquid molecules (particles). From special point of view, it can be seen as a finite difference stencil for solving the Boltzmann transport equation (BTE) [1]. Moreover, the incompressible Navier-Stokes equations can be derived from BTE by LBM discretization with a proper choice of the collision operator [2]. In this article, we first introduce this method and base description for its implementation. In numerical examples section, the validity of this method is confirmed by comparing the numerical solution to the exact solution of the steady-state plane lid-driven cavity flow and convergence of solution is established. Interesting numerical example describing flow past a circular cylinder is also carried out.

2. Lattice Boltzmann model

The principle of the Lattice Boltzmann method is based on the Boltzmann's theory of gases. Fundamental imagination describes the fluid as the matter consisting of a large number of random moving particles. The momentum exchange between particle sets is achieved through the streaming and collision operators. The numerical solution of the Boltzmann transport equation, which is defined as [1]

$$\frac{\partial f}{\partial t} + u \cdot \nabla f = \Omega, \quad (1)$$

where $f(x, t)$ represents the particle distribution, u is the velocity, and Ω describes the collision operator model that can be used to represent this inter-particle transfer of momentum. The LB method simplifies Boltzmann's original idea of liquid dynamics by reducing the number of particles and restricting their position into the nodes of a lattice or stencil [3]. For a two-dimensional model, a particle can be streamed in a nine directions, including the one staying at rest. These velocities are referred to as the mesoscopic velocities and denoted by e_i

$$e_i = \begin{cases} (0,0) & i = 0 \\ (1,0), (0,1), (-1,0), (0,-1) & i = 1,2,3,4 \\ (1,1), (-1,1), (-1,-1), (1,-1) & i = 5,6,7,8 \end{cases} \quad (2)$$

where $i = 0, \dots, 8$. This lattice model is commonly known as the D2Q9 model [4] as it is two-dimensional and involves vector of nine components. Figure 1 shows a typical node of two-dimensional model with nine velocities e_i . For each particle on the lattice grid, we associate a discrete probability, real number based (in contrast to lattice gas automata method) distribution function $f_i(x, e_i, t)$ which describes the probability of streaming in possible lattice directions.

Fig.1: D2Q9 lattice model illustration.

The macroscopic fluid density is defined as a summation of mesoscopic particle distribution function [1, 2]

$$\rho(x, t) = \sum_{i=0}^8 f_i(x, t). \quad (3)$$

Accordingly, the macroscopic velocity field $u(x, t)$ is represented as an average of mesoscopic velocities e_i with influence altered by the values of distribution function f_i

$$u(x, t) = \frac{1}{\rho} \sum_{i=0}^8 c f_i e_i. \quad (4)$$

The procedures for the collision and streaming processes representing the basal steps in LBM algorithm are given by

$$f_i(x + c e_i \Delta t, t + \Delta t) - f_i(x, t) = -\frac{f_i(x, t) - f_i^{eq}(x, t)}{\tau}. \quad (5)$$

In the proposed implementation of the LB numerical model, streaming and collision are computed separately and extra attention is given to these when dealing lattice nodes at the boundary. Figure 2 shows graphically how the streaming step procedure for the interior nodes.

Fig. 2: Distribution function streaming operation for D2Q9 lattice model.

In the collision term of $f_i^{eq}(x,t)$ is the equilibrium distribution, and τ is considered as the relaxation time towards local equilibrium. For simulating single phase flows, it suffices to use Bhatnagar-Gross-Krook (BGK) collision model. Main part of BGK model is the equilibrium distribution defined as [1]

$$f_i^{eq}(x,t) = w_i \rho + \rho s_i(u(x,t)), \quad (6)$$

where $s_i(u)$ is defined as follows:

$$s_i(u) = w_i \left[3 \frac{e_i u}{c} + \frac{9}{2} \frac{(e_i u)^2}{c^2} - \frac{3}{2} \frac{u u}{c^2} \right], \quad (7)$$

where w_i are lattice weights.

The solution algorithm can be described as follows [4]:

1. Initialize macroscopic properties ρ , u and distribution values f_i and f_i^{eq} .
2. Stream distribution f_i to neighbours nodes.
3. Evaluate macroscopic properties ρ and u using distribution function values.
4. Use equation (6) to evaluate f_i^{eq} .
5. Model collision process using BGK to obtain the updated distribution function f_i .
6. While not end of simulation, return to step 2.

During the streaming and collision process of the particles, the nodes that serve as global domain boundary require some special numerical procedures for the distribution functions evaluation in order to satisfy the macroscopic boundary conditions. This boundary condition in this article is non-slip boundary condition (BC) and boundary conditions representing prescribed velocity and pressure BC [3].

3. Lid-driven cavity flow

In this simulation we have a 2D fluid flow that is driven by a lid at the top which moves at a speed of $u = 1.0$ in the right direction. The other three walls have no-slip boundary conditions for the velocity, $u = 0$ and $v = 0$. The initial state is described by the zero velocity field and the initial values of the distribution function is determined using the weights $f_i = w_i$.

Fig. 3: Lid-driven cavity, streamlines pattern for different Reynolds numbers.

This results in an initial condition that $\rho = 1$ from (3). The only exception is the velocity of the fluid on the top is set to be $u = 1.0$. Simulations were done with a 600×600 lattice grid with $Re = 400$ and 1000 .

Fig. 4: Lid-driven cavity, velocity magnitude for different Reynolds numbers.

Fig. 5: Lid-driven cavity, $Re = 1000$, vertical velocity components along horizontal line $y = 0.5$ and horizontal velocity components along vertical line $x = 0.5$.

The results of numerical simulation were compared with the results presented in [5], see Fig. 5.

4. Flow past a circular cylinder

The study of flow past an object dates its history back to ships design. Many investigators were interested in the new ship shapes and describing the behaviour of the flow past the ship body. One of the most important results of these investigations is that Reynolds number plays an important role in characterizing the behaviour of the flow.

Fig. 6: Flow past cylinder, streamlines pattern.

Fig. 7: Flow past cylinder, velocity magnitude contours.

This numerical study consists of the 2D channel with a steady Poiseuille flow profile, with circular cylinder immersed into the flow at the start of the simulation. On lattice grid the no-slip boundary conditions applied to the boundary of the cylinder as well as the top and the bottom boundaries of the channel. Velocity and pressure (density) boundary conditions are applied to the inlet and the outlet side of the channel.

Fig. 8: Flow past cylinder, vorticity contours - Karman vortex street.

Numerical simulation with $Re = 400$ have been carried out on the 2000 x 400 lattice grid and the results (Figs. 6, 7, 8) shows the generation of the Karman vortex street in the area past the cylinder (Fig. 8).

5. Conclusion

In this project the application of the Lattice Boltzmann method was presented. The validity of this method was verified by comparing numerical solutions to the solutions of other researchers.

Two nontrivial simulations: lid-driven cavity flow and flow past a circular cylinder were performed and they agreed closely with physical situations.

The Lattice Boltzmann method has several advantages for fluid simulations over traditional numerical methods used for direct solution of the Navier-Stokes equations. Complex boundaries are much easier to deal with using on-grid bounce-back boundary condition and thus LBM can be applied to simulate flows with complex geometries such as porous media flows. LBM can be easily parallelized because of its local nature and therefore is one of the best candidates for the numerical methods for massive parallelization.

References

- [1] GUO, Z. - SHI, B. - WANG, N.: Lattice BGK Model for Incompressible Navier-Stokes Equation. In: Journal of Computational Physics. ISSN 0021-9991, 2000, Vol. 165, p. 288 - 306.
- [2] SUKOP, M. - THORNE, D.T.: Lattice Boltzmann Modelling: an introduction for geoscientists and engineers. Springer Verlag, 1st edition, 2006.
- [3] BEGUM, R. - BASIT, M.A.: Lattice Boltzmann Method and its Applications to Fluid Flow Problems. In: European Journal of Scientific Research. ISSN 1450-216X, 2008, Vol. 22, p. 216 - 231.
- [4] SUCCI, S.: The Lattice Boltzmann Equation for Fluid Dynamics and Beyond. Oxford University Press, Oxford, 2001, p. 82 - 84.
- [5] GHIA, U. - GHIA, K. N. - SHIN, C. T.: High-resolutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method. In: Journal of Computational Physics. ISSN 0021-9991, 1982, Vol. 48, p. 387 - 411.