Title	実数型格子ガス法による熱流動解析に関する研究
Author(s)	今川, 洋造
Citation	
Issue Date	2001-03
Type	Thesis or Dissertation
Text version	author
URL	http://hdl.handle.net/10119/1478
Rights	
Description	Supervisor:松澤 照男, 情報科学研究科, 修士

A heat flow simulation by the continuous-velocity lattice-gas model

Youzou Imagawa

School of Information Science,
Japan Advanced Institute of Science and Technology

February 15, 2001

Keywords: lattice-gas model, continuous-velocity lattice-gas model, Maxwell-Boltzmann distribution, Benard convection.

Background and Purpose

In general numerical methods, we obtain the numerical solusion by solving the systems (Navier-Stokes equation,etc.) which govern the fluid flow. As another analysis method, there are molecular methods to analyze the fluid dynamics by simulating the movement of molecular which constructs the fluid material. These methods model the macroscopic fluid dynamics by microscopic molecular movement.

If we completely simulate the fluid flow by using the molecular method, a huge number of molecular is needed. It is necessary to simulate the fluid dynamics by the movement of sample molecular.

The lattice gas model is one of the molecular method to analyze the fluid flow.

This model has the following property:

- Particle position, velocity, space, and time are discrete.
- Particle movement consists of two section, streaming section and collision section. Particle moves at its velocity per unit time in streaming section. Particles change their momentum and velocity in collision section.
- Collision occurs on lattice point. Collision rule is expressed by Boolean operation.
- Local physical value is calculated by spacial average, according to circumstances, time average.

- The distribution of velocity at equilibrium state is Fermi-Dirac distribution.

However, in lattice-gas model, it is necessary to establish the collision rule and lattice shape to satisfy the isotropy of particle movement. Moreover, because the collision rule is expressed by boolean operator, exclusive rule must be applied, and the number of particles on lattice point is limited. And when we make a space discrete, it is necessary to use the hexagonal lattice (which is called FHP lattice) instead of square lattice. And, because the number of state of particle momentum is very few, we cannot simulate the thermal flow by using lattice gas model.

Recently,continuous-velocity lattice-gas model was developed. In this model, space and time are discrete. Discrete space is composed of square lattices. However, particle velocity is real number. As particle collision occurs on lattice point, particle position in real number is transferred at lattice point by using the probability process which is decided by particle position and velocity. On particle collision, particles on the lattice point exchange their momentum and velocity by rotating each vectors which indicate the difference of the velocity of centre of mass of the colliding particles. By this collision rule, momentum and kinetic energy of particles on the lattice point are conserved, and there is no limit of the number of particles on lattice point. Moreover, the distribution of velocity at equilibrium state is Maxwell-Boltzmann distribution which is similar to general particle movement. So, it may be possible to simulate the more general physical phenomenon by continuous-velocity lattice-gas model than by lattice-gas model.

There is no limit of the number of state of particle momentum and energy equation is produced in this model. It is supposed that we can analyze the heat flow phenomenon by continuous-velocity lattice-gas model as we set the suitable boundary condition.

However, the research for concrete computation of heat flow phenomenon by continuous-velocity lattice gas model has not still studied. This research aims at the simulation and observation of heat flow problem by the continuous-velocity lattice-gas model.

Result

I simulated the 2D couette flow, and it was realized that the boundary condition which eliminates the parallel and vertical components of particle velocity was expressed by the non-slip boundary condition.

I simulated the 2D cavity flow, and I compared the result by this model with numerical solution. It was realized that this model is available for the flow of hundreds of Reynolds number. It was confirmed that 2D thermal cavity flow could be simulated and boundary condition with temperature property was suitable.

I simulated the 2D Benard convection, it was realized thad by the influences of gravity and thermal boundary condition, thermal convection generates, and temperature and density were transported by thermal convection.

Problem to be solved

As the speed of sound and viscosity coefficient are proportional to temperature, it is necessary to increase the lattice number for simulating the flow of large Reynolds number. And, continuous-velocity lattice-gas model has excellent property that it is easy to extend to the 3D flow simulation. However, 3D flow simulation needs much number of lattice point and particles than 2D flow simulation. In continuous-velocity lattice-gas model, because computation time increases in proportion to the number of lattice points and particles, I consider that it needs much more time to simulate the 3D flow. However, because the computation of this model is explicit, we consider that it is possible to reduce the computation time by the parallel computation.

The square lattice has been used in continuous-velocity lattice-gas model. In this research, computation space is square or rectangle space. However, actually, the computation of fluid flow around the complicated object is needed. In lattice-gas model, curved surfaces and curved lines are expressed by using the fine lattice. However, computation cost is huge. We should consider how we set the boundary condition on the complicated surface.