Computers & Fluids 80 (2013) 453–458

Contents lists available at SciVerse ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

Simulation of turbulent flow by lattice Boltzmann method and conventional method on a GPU

I. Tanno [a,*], T. Hashimoto [b], T. Yasuda [c], Y. Tanaka [d], K. Morinishi [e], N. Satofuka [e]

[a] Tsukuba University of Technology, Amakubo, Tsukuba 305-8520, Japan
[b] Kinki University, Kowakae 3-4-1, Higashiosaka, Osaka 577-8502, Japan
[c] The University of Shiga Prefecture, Hassaka 2500, Hikone, Shiga 522-8533, Japan
[d] Toyo Tire & Rubber Co. Ltd., Fujinoki 2-2-13, Itami, Hyogo 644-0847, Japan
[e] Kyoto Institute of Technology, Goshokaidocho Matsugasaki Sakyo-ku, Kyoto 606-8525, Japan

ARTICLE INFO

Article history:
Received 30 September 2011
Received in revised form 17 January 2012
Accepted 22 January 2012
Available online 9 February 2012

Keywords:
Lattice Boltzmann method
Artificial compressibility method
Pseudospectral method
Homogeneous isotropic turbulence

ABSTRACT

In this study, the lattice Boltzmann method, pseudospectral method, and artificial compressibility method were implemented on both CPU and GPU machines. Homogeneous isotropic turbulent flows were calculated using these three methods with the C language and CUDA library. The computational results show that the flow field obtained by the lattice Boltzmann method was almost the same as that obtained by the pseudospectral method. Among these three methods, the computational time of the lattice Boltzmann method on a GPU was the shortest of all calculations. Thus, the lattice Boltzmann method was well accelerated by GPUs. These results proved that the lattice Boltzmann method on a GPU has advantages of accuracy and computational speed.

1. Introduction

Recently, an increasing number of reports concerning numerical simulations performed on GPUs have been presented [1,2]. Previously, when computer simulations were tried on GPUs, knowledge about computer graphics was required. Now, the CUDA library developed by NVIDIA allows programs for computer simulations on GPUs to be created without knowledge about computer graphics.

Because a single GPU has a huge amount of arithmetic units and a wide-bandwidth data path between the GPU processor and memory, a GPU has arithmetic performance that is almost 30–100 times greater than that of a CPU. Extra devices, such as the ClearSpeed devices, to increase arithmetic performance for computer simulation have been proposed, but they have not been used in mainstream computer simulations because of their high cost. Since GPUs are general-purpose devices, whose prices are reasonable, and consume relatively little electricity, recent supercomputers have tended to use a large number of GPUs to increase their performance. Nowadays, simulations on GPUs are thus one of the hot topics in computational fluid dynamics (CFD). A single GPU, which has many arithmetic units, can be regarded as a kind of massively parallel computer. In general, computational schemes suitable for parallel computers are suitable for GPUs as well. Although the Navier–Stokes equations have traditionally been employed as CFD algorithms, study on the lattice Boltzmann method (LBM) [3], which is a relatively new CFD algorithm, is also one of the hot topics in CFD. It has been derived that the lattice Boltzmann equations and the Navier–Stokes equations give the same flow fields when the Mach number is low enough. Therefore, the LBM is usually used to simulate incompressible flows. Although many methods to solve the incompressible Navier–Stokes equation have been proposed, in almost all of these methods it is necessary to solve the Poisson equation, which requires an enormous number of iterative calculations to obtain the pressure fields. By contrast, since the D2Q9-type lattice Boltzmann method only requires calculations of the lattice Boltzmann equations at each of nine speeds, the amount of computations of the LBM tends to be smaller than that of the traditional method. Although the LBM has second-order accuracy for space derivative terms and first-order accuracy for time derivative terms, in practice this method gives higher accuracy. Since the Courant number of the LBM is always 1, numerical viscosity, which is produced by an upwind scheme for stability, will not appear. Satofuka and Nishioka [4] calculated homogeneous isotopic turbulent flows and compared the results obtained by the LBM and by the 10th-order finite difference method, showing that the LBM has good accuracy and is suitable for parallel computers. Also Satofuka and Nishida [5], and Satofuka et al. [6] reported that results by 10th order scheme agree with that by pseudospectral method. These reports indirectly show that the accuracy of LBM is quite close to that of pseudospectral method.

* Corresponding author. Tel.: +81 29 858 9367.
E-mail address: itaru_t@a.tsukuba-tech.ac.jp (I. Tanno).

0045-7930/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
doi:10.1016/j.compfluid.2012.01.011

The spectral method [7] or pseudospectral method [8] (PS) is usually employed for direct numerical simulations (DNS) of turbulent flows. These methods provide accurate solutions even if the number of grid points is relatively small. Since both the spectral method and PS require Fourier transforms, these methods are not suitable for parallel computers. These methods work very well for flow fields that have periodic boundary conditions, and they are thus one of the important solutions in CFD. However, it is difficult for these methods to simulate the flow fields that appear inside fluid machinery or in industrial applications, because it is difficult to impose wall conditions or other boundary conditions with these methods.

Artificial compressibility method (ACM) [9] is one method to simulate incompressible flow. Although unlike other methods based on the Navier–Stokes equations, ACM does not calculate a pressure Poisson equation, ACM needs subiterations to impose the continuity conditions for unsteady simulations. However, Ohwada and Asinari [10] reported that ACM can provide unsteady flow fields without subiterations. If subiterations are removed from ACM, the computational costs will be significantly reduced.

Industrial applications are important subjects of CFD. To treat these applications, the CFD method must have the following three characteristics:

(1) The CFD method can accurately simulate turbulent flow.
(2) The CFD method can impose many types of boundary conditions without special effort.
(3) The CFD method can be rapidly and easily run on massively parallel computers.

The LBM running on GPUs can satisfy these three characteristics. In this paper, we calculated a homogeneous isotropic turbulent flow using the LBM, PS, and ACM without subiteration.

These calculations were performed on a CPU and GPUs. The computational times and accuracy of the calculations and were compared among these methods, showing that the LBM on a GPU is a relatively accurate and fast method, making it suitable for practical use.

2. Numerical methodology

2.1. The lattice Boltzmann method

The nine-velocity square lattice model was used. Fig. 1 shows the relationships among the grid nodes. One grid node has eight connections to its eight neighboring grid points. Because particles on a grid point will stay on that point or move to the neighboring grid points within a time unit, there are three particle speeds. The speed of particles that move along the x or y axis is $|\boldsymbol{e}_{1i}| = 1$, and that of particles that move in a diagonal direction is $|\boldsymbol{e}_{2i}| = \sqrt{2}$. The lattice Boltzmann equation is

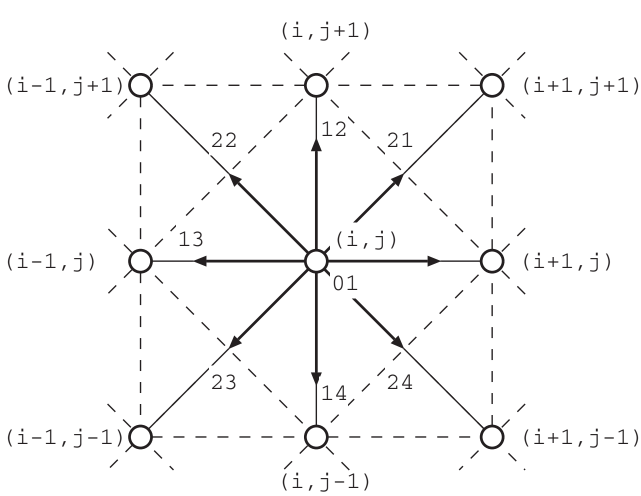

Fig. 1. Square lattice model and velocity of particles.

$$f_{\sigma i}(\boldsymbol{x}+\boldsymbol{e}_{\sigma i}, t+1) - f_{\sigma i}(\boldsymbol{x}, t) = \Omega_{\sigma i} \tag{1}$$

where $f_{\sigma i}$, x, $\boldsymbol{e}$, t, and $\Omega_{\sigma i}$ are the distribution function, position vector of the nodes, velocity of the particles, time, and collision operator, respectively. In order to reduce the complexity of the collision operator, the single-time relaxation approximation proposed by Bhatnagar et al. [11], was applied to Eq. (1). Then, the lattice Boltzmann BGK (LBGK) equation was derived as

$$f_{\sigma i}(\boldsymbol{x}+\boldsymbol{e}_{\sigma i}, t+1) - f_{\sigma i}(\boldsymbol{x}, t) = \frac{1}{\tau}\left[f_{\sigma i}(\boldsymbol{x}, t) - f_{\sigma i}^{(0)}(\boldsymbol{x}, t)\right] \tag{2}$$

Here $f_{\sigma i}^{(0)}$ indicates the equilibrium distribution functions defined as

$$f_{01}^{(0)} = \rho\alpha - \frac{2}{3}\rho\boldsymbol{u}^2 \tag{3}$$

$$f_{1i}^{(0)} = \rho\beta + \frac{1}{3}\rho(\boldsymbol{e}_{1i}\cdot\boldsymbol{u}) + \frac{1}{2}\rho(\boldsymbol{e}_{1i}\cdot\boldsymbol{u})^2 - \frac{1}{6}\rho\boldsymbol{u}^2 \tag{4}$$

$$f_{2i}^{(0)} = \rho\frac{(1-4\beta-\alpha)}{4} + \frac{1}{12}\rho(\boldsymbol{e}_{2i}\cdot\boldsymbol{u}) + \frac{1}{8}\rho(\boldsymbol{e}_{2i}\cdot\boldsymbol{u})^2 - \frac{1}{24}\rho\boldsymbol{u}^2 \tag{5}$$

$$\alpha = 4/9 \tag{6}$$

$$\beta = 1/9 \tag{7}$$

and τ is the relaxation time defined as

$$\tau = \frac{6\nu+1}{2} \tag{8}$$

Here ν is the kinematic viscosity. In order to obtain the density per node ρ and macroscopic velocity $\boldsymbol{u}$, the following equation was used:

$$\rho = \sum_{\sigma}\sum_{i} f_{\sigma i} \tag{9}$$

$$\rho\boldsymbol{u} = \sum_{\sigma}\sum_{i} f_{\sigma i}\boldsymbol{e}_{\sigma i} \tag{10}$$

The time step is chosen as $\Delta t = 1.0$.

2.2. The traditional method

The incompressible Navier–Stokes equations are

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{11}$$

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + \frac{\partial p}{\partial x} = \frac{1}{\mathrm{Re}}\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial x^2}\right] \tag{12}$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + \frac{\partial p}{\partial y} = \frac{1}{\mathrm{Re}}\left[\frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial y^2}\right] \tag{13}$$

In this study, the PS and ACM were used in order to numerically solve these equations.

2.3. The pseudospectral method

Generally, the spectral method and PS are employed for DNS of turbulent flows, because these methods can provide accurate solutions even if the number of grid points is relatively small. In this