Development of a Fluid Simulation Approach by Massively Parallel Bit-wise Operations with a New Viscosity Control Method

Project Representative
Hiroshi Matsuoka
Research Institute of Electrical Communication, Tohoku University

Authors
Hiroshi Matsuoka*1, Noriko Kikuchi*1,2, Tadashi Watanabe*3, Mitsuo Yokokawa*4, Shin-ichi Mineo*4, Ken-ichi Itakura*5, Yukio Iwaya*1, Yukio Fujinawa*6 and Yoshihiko Orita*7

*1 Research Institute of Electric Communication, Tohoku University
*2 Customer System Company
*3 Japan Atomic Energy Agency
*4 Advanced Institute for Computational Science, RIKEN
*5 Earth Simulator Center, Japan Agency for Marine-Earth Science and Technology
*6 Fujinawa Earthquake Research & Development
*7 Japan Marine Science Fundation

The purpose of this project is to simulate the three-dimensional vortices from a circular cylinder in the fluid-flow by using a massively parallel Lattice Gas Method, which is expected to provide excellent computing performance on a large-scale vector computer, and to evaluate the applicability of the method to similar large-scale fluid-simulation problems by comparing the results with those of experiments or numerical calculations by solving Navier-Stokes equations. In particular, we try to realize the fluid-flow simulation at high Reynolds numbers by using a new viscosity-control method that we call “multi-stage collisions of two particles”. In case of the method, propagation of particles occurs after several times of collisions of randomly selected two particles at each node. By using the method, we can calculate fluid-flow at somewhat higher Reynolds number without increasing the number of nodes of lattice. Another feature of our method is “massively parallel bit-wise calculations”. We obtained the value of 96.8% as the calculation-efficiency by using 1024 vector CPUs of Earth Simulator. We regard that the value is a good performance.

Keywords: lattice gas automaton, fluid dynamics, massively parallel computing, vortex from a cylinder, vector computers

1. Purpose of the project
The purpose of this project is to simulate the three-dimensional vortices from a circular cylinder in the fluid-flow by using a massively parallel Lattice Gas Method, which is expected to provide excellent computing performance on a large scale vector computer, and to evaluate the applicability of the method by comparing the results with those of experiments or numerical calculations by solving Navier-Stokes equations.

In particular, we try to realize the fluid-flow simulation at high Reynolds numbers by using a new viscosity-control method without increasing the number of nodes of lattice.

2. Plan of three-year research
We are trying to complete the following three subjects in three years.

(1) Improvement of the calculation-efficiency of massively parallel bit-wise operation method in the FY 2010
(2) Confirmation on the simulation results of vortices shedding from a circular cylinder of finite length in the FYs 2010 and 2011
(3) Simulation of the fluid-flow at high Reynolds numbers by the new viscosity-control method in the FYs 2011 and 2012.

3. Calculation method
3.1 Background of the calculation method
According to the previous study[1], it is known that wind or water tunnels can be indifferently used for testing low Mach number flows, provided the Reynolds numbers are identical. Indeed, two fluids with quite different microscopic structures can have the same macroscopic behavior because the form of the macroscopic equations is entirely governed by the microscopic conservation laws and symmetries. Such observations have
led to a new simulation strategy for fluid dynamics: fictitious micro-world models obeying discrete cellular automata rules have been found, such that two- and three-dimensional fluid dynamics are recovered in the macroscopic limit. The class of cellular automata used for the simulation of fluid dynamics is called “lattice gas models”, and many lattice gas models have been proposed.

In our study, we use one-speed models for the simulation of fluid dynamics. The relevant aspects of the models are as follows: there is a regular lattice, the nodes of which are connected to nearest neighbors through links of equal length; all velocity directions are in some sense equivalent and the velocity set is invariant under reversal; at each node there is a cell associated with each possible velocity.

Each node can be occupied by one particle at most; particles are indistinguishable; particles are marched forward in time by successively applying collision and propagation rules; collisions are purely local, having the same invariances as the velocity set; and collisions conserve only mass and momentum.

3.2 FCHC model for three-dimensional simulation

In order to simulate three-dimensional vortices shedding from a circular cylinder, we selected a face-centered-hypercubic (FCHC) model among from several one-speed models. The FCHC model is a four-dimensional model introduced by d’Humières, Lallemand, and Frisch in 1986[2]. Three dimensional regular lattices do not have enough symmetry to ensure macroscopic isotropy. The detailed FCHC model that we use in this study and the schematic diagram of simulated flow are explained in the following three figures.

As shown in Fig. 1, the coordinate X is the direction of flow, and the coordinate Z is parallel to the circular cylinder. A position of (X, Y, Z) represents the position of each cell. Every cell contains 32 nodes as depicted in Fig.1 Each node exists in the four-dimensional space. The fourth coordinate R is represented by the radius of sphere at each three-dimensional position.

As shown in Fig. 2, particles can have 24 kinds of velocities, that is, (∆X, ∆Y, ∆Z, ∆R) = (±1, ±1,0,0), (±1,0, ±1,0), (±1,0,0, ±1), (0, ±1, ±1,0), (0, ±1,0, ±1) or (0,0, ±1, ±1), and the magnitude of the velocities is equal to $\sqrt{2}$, when the interval between two nearest nodes has a unit length.

Many particles propagate from node to node and make a collision at each node. The rules of propagation and collision are presented in Fig. 2 and Fig. 3, respectively.

The features of our method are “massively parallel bit-wise calculations” and “multi-stage collisions of two particles”.

Bit-wise parallel calculations are realized by vector operations on the arrangement representing the state of a cell. The arrangement is given by the form of 4-dimensional integer arrangement bit[D][Z][Y][X] that has 32 elements with the value of “1” or “0”. If the k-th bit of the arrangement bit[D][Z][Y][X] equals to “1”, this means that a particle moving toward the direction D exists at the k-th node of the cell locating at (X, Y, Z). “1” or “0” means existence or nonexistence of a particle, respectively.

Multi-stage collisions of randomly selected two particles at each node are useful for making a smaller correlation between fluid-velocities of two different cells a little apart from each other. This means that the fluid has smaller viscosity and...
simulation at higher Reynolds numbers becomes somewhat easier.

3.3 Case study for the simulation of vortices shedding from a circular cylinder of infinite length

We numerically simulated the vortices shedding from a circular cylinder of infinite length.

Figure 4 shows the results of transient simulation of the fluid-momentum on the plane that Z equals to a certain constant value. General feature of the transient from twin vortices to Karman whirlpools is conceptually equal to those of the experiments by Taneda[3].

The number of nodes for the calculation is 3072(X)×768(Y)×768(Z)×4(R) in four dimensional space.

Propagation rules of a particle

1. A particle staying at the node (X, Y, Z, R) with the velocity (ΔX, ΔY, ΔZ, ΔR) at the time T, will propagate to the next node (X+ΔX, Y+ΔY, Z+ΔZ, R+ΔR) at the time T+1 and have the same velocity when the next node is liquid.

2. A particle staying at the node (X, Y, Z, R) with the velocity (ΔX, ΔY, ΔZ, ΔR) at the time T, will have the reversed velocity (-ΔX, -ΔY, -ΔZ, -ΔR) at the same node at the time T+1 when the next node is solid.

Collision rules of particles

1. The number of particles and the sum of their momentums for each direction are conserved.

2. At most one particle can move to the same direction at a time.
Figure 5 shows the effects of the number of multi-stage collisions of two particles on transient simulation of the flow past a circular cylinder of infinite length[4]. Increase of the number of multi-stage collisions from 5 to 90 makes higher Reynolds numbers so that Karman whirlpools occur.

3.4 Results of the study in FY 2011
Regarding the improvement of the calculation-efficiency of “massively parallel bit-wise calculations”, we obtained the value of 96.8% by using 1024 vector CPUs of Earth Simulator. We regard that the value is a fairly good performance.

In FY 2011, we simulated many cases of the vortices shedding from a circular cylinder of finite length. One of the results is shown in Fig. 6. Further study is needed in FY 2012.

Acknowledgement
We greatly appreciate JAMSTEC for giving us a chance of using Earth Simulator and very kind support for our study.
References


新粘性制御法による超並列ビット演算流体シミュレーション手法の開発

プロジェクト責任者
松岡 浩 東北大学 電気通信研究所
著者
松岡 浩*1，菊池 範子*1，2，渡辺 正*3，横川三津男*4，峰尾 真一*4，板倉 憲一*5
岩谷 幸雄*1，藤縄 幸雄*6，折田 義彦*7
*1 東北大学 電気通信研究所
*2 カストマシステム株式会社
*3 日本原子力研究開発機構
*4 理化学研究所 計算科学研究機構
*5 海洋研究開発機構 地球シミュレータセンター
*6 藤縄地震研究所
*7 日本海洋科学振興財団

本プロジェクトの目的は、大規模ベクトル計算機において優れた計算性能を発揮することが期待される格子ガス法超並列計算法を用いて、流体流れの中に置かれた円柱後流に生じる3次元渦のシミュレーションを行い、その結果を実験やナビエ・ストークス方程式を解く数値計算の結果と比較することによって、同様な大規模流体シミュレーション問題への本手法の適用可能性を評価することにある。

特に、“多段2体粒子衝突”と呼ぶ新しい粘性制御法を用いて、高いレイノルズ数領域における流体流れのシミュレーションの実現を図る。この手法では、各ノードにおいて、ランダムに選択された2粒子が数10回衝突を起こしたあとに、粒子の並進移動を行う。本手法を用いると、格子点の数を増やすことなく、ある程度高いレイノルズ数領域の流体流れを計算することができる。本手法を用いて、有限長円柱の後流に生じる3次元渦のシミュレーションを行った。

また、我々が提案する手法のもうひとつの特徴は、“超並列ビット計算”である。地球シミュレータの1024個のベクトルCPUを用いて、並列化効率96.8%というかなり良い値を得た。

キーワード：格子ガスオートマトン, 流体力学, 超並列計算, 円柱後流渦, ベクトル計算機