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

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

Authors
Hiroshi Matsuoka Tohoku University
Noriko Kikuchi Science Services Company
Tadashi Watanabe Fukui University
Mitsuo Yokokawa Kobe University
Shin-ichi Mineo Research Organization for Information Science and Technology
Ken-ichi Itakura Earth Simulator Center, Japan Agency for Marine-Earth Science and Technology
Yukio Iwaya Tohoku University
Yukio Fujinawa Fujinawa Earthquake Research & Development
Yoshihiko Orita 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 (NS) 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 many times of collisions of randomly selected two particles at each node. In addition, in the study of FY 2012, three speeds fifty-four velocities model was introduced. The model was shown by Teixeira in his Ph. D thesis[3], 1992, and it can make the mean free path of particles less than 1 lattice spacing. By using the method, we can simulate fluid-flow at somewhat higher Reynolds number without increasing the number of nodes of lattice. The other features of the method are "massively parallel bit-wise calculations" and "correct reproduction of the macroscopic behavior of the continuity and NS equations".

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. Calculation method

2.1 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 in FYs 2010 and 2011, we used one speed twenty-four velocities model 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.

2.2 FCHC three speeds fifty-four velocities 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. 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. In 1992, Teixeira shows in his Ph. D thesis[3] at MIT that FCHC three speeds fifty-four velocities model can make the mean free path of particles less than 1 lattice spacing and reproduce macroscopically the behavior of the continuity and Navier-Stokes (NS) equations".

The detailed FCHC model that we use in this study and the schematic diagram of simulated flow are explained in the following two 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 three speeds fifty-four velocities at each node, that is;

Speed 0-Group: one velocity but six particles

(ΔX, ΔY, ΔZ, ΔR) = (0, 0, 0, 0)

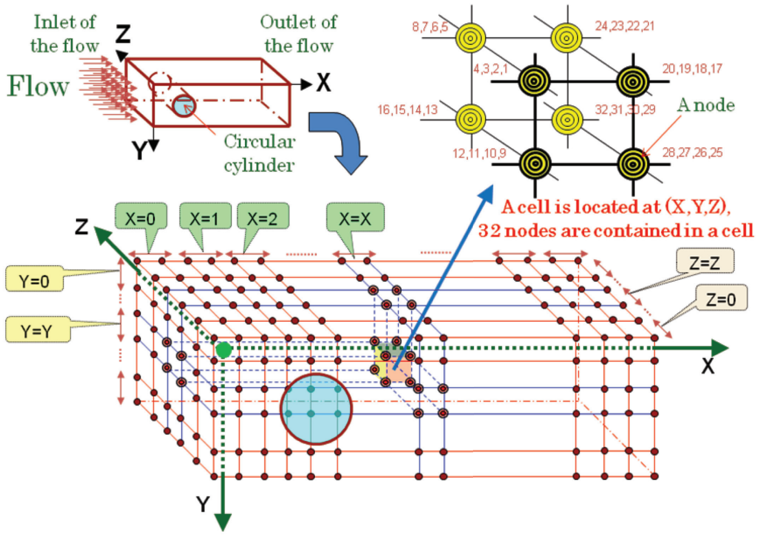
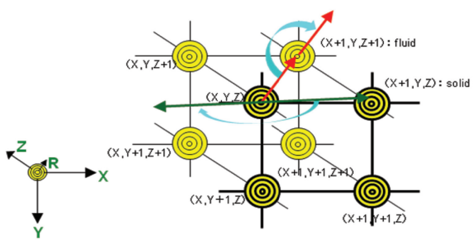


Fig. 1 Cells and nodes for simulating the three dimensional vortices shedding from a circular cylinder.

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.



(⊿X,⊿Y,⊿Z,⊿R)	(⊿X,⊿Y,⊿Z,⊿R)
1(0 0 -1 -1)	25(-1 -1 -1 -1)
2(0 0 -1 1)	26(-1 -1 -1 1)
3(0 -1 0 -1)	27(-1 -1 1 -1)
4(0 -1 0 1)	28(-1 -1 1 1)
5(0 -1 -1 0)	29(-1 1 -1 -1)
6(0 -1 1 0)	30(-1 1 -1 -1)
7(-1 0 0 -1)	31(-1 1 1 -1)
8(-1 0 0 1)	32(-1 1 1 1)
9(-1 0 -1 0)	33(1 -1 -1 -1)
10(-1 0 1 0)	34(1 -1 -1 1)
11(-1 -1 0 0)	35(1 -1 1 -1)
12(-1 1 0 0)	36(1 -1 1 1)
13(1 -1 0 0)	37(1 1 -1 -1)
14(1 1 0 0)	38(1 1 -1 1)
15(1 0 -1 0)	39(1 1 1 -1)
16(1 0 1 0)	40(1 1 1 1)
17(1 0 0 -1)	41(2 0 0 0)
18(1 0 0 1)	42(0 2 0 0)
19(0 1 -1 0)	43(0 0 2 0)
20(0 1 1 0)	44(0 0 0 2)
21(0 1 0 -1)	45(-2 0 0 0)
22(0 1 0 1)	46(0 -2 0 0)
23(0 0 1 -1)	47(0 0 -2 0)
24(0 0 1 1)	48(0 0 0 -2)
49~54 (⊿X,⊿Y,⊿Z,⊿R) =(0 0 0 0)	

Fig. 2 Propagation rules from node to node.

Speed √2-Group: 24 velocities

(ΔX, ΔY, ΔZ, ΔR) = (±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)

Speed 2-Group: 24 particles

(ΔX ΔY, ΔZ, ΔR) = (±1, ±1, ±1, ±1), (±2, 0, 0, 0), (0, ±2, 0, 0), (0, 0, ±2, 0) or (0, 0, 0, ±2)

Many particles propagate from node to node and make a collision at each node.

The rules of propagation are presented in Fig. 2.

The rules of collisions at each node are given as follows:

1. The number of particles, the sum of their momentums for each direction, and the sum of their energies are conserved.
2. At most one particle can have the same velocity at a time.

The features of our method are "massively parallel bit-wise calculations", "multi-stage collisions of two particles" and "correct reproduction of the macroscopic behavior of the continuity and NS equations".

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.

Teixeira showed[3] that FCHC three speeds fifty-four velocities model can make the mean free path of particles less than 1 lattice spacing. Application of multi-stage collisions[4] of randomly selected two particles at each node is useful for making further 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. Case-study simulations

3.1 Case studies 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 3 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

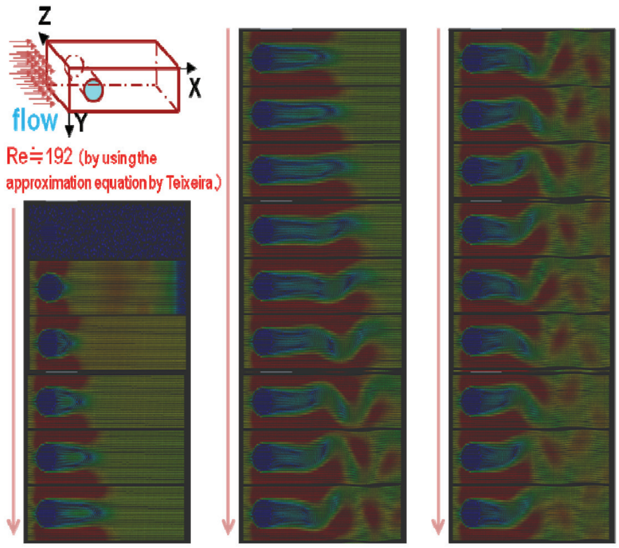


Fig. 3 A transient simulation result of the flow past a circular cylinder of infinite length.

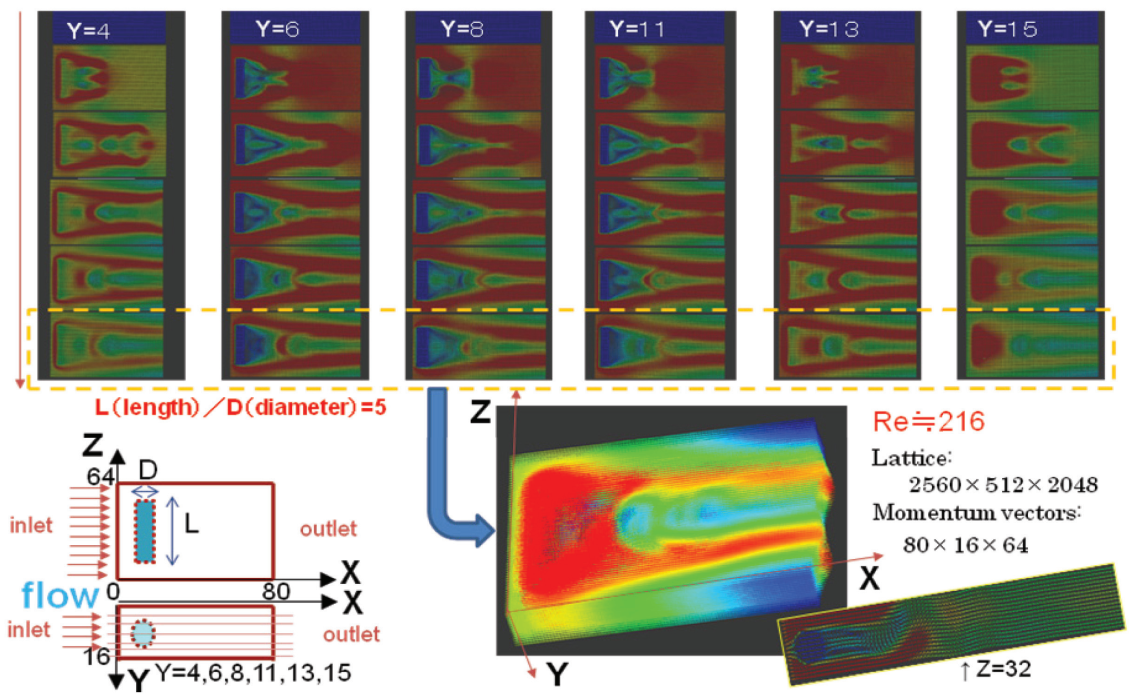


Fig. 4 A transient simulation result of the flow past a circular cylinder of finite length.

to Karman whirlpools is conceptually equal to those of the experiments by Taneda[5].

The number of nodes for the calculation is 1536(X) × 384(Y) × 384(Z) × 4(R) in four- dimensional space.

3.2 Case studies for the simulation of vortices shedding from a circular cylinder of finite length

We simulated many cases of the vortices shedding from a circular cylinder of finite length. One of the results calculated by 1024 CPUs of ES2 is shown in Fig. 4.

Acknowledgement

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

References

[1] U. Frisch, D. d'Humières, B. Hasslacher, P. Lallemand, Y. Pomeau, and J-P. Rivet, "Lattice Gas Hydrodynamics in Two and Three Dimensions", Complex Systems, 1, pp.649-707, 1987.

[2] D. d'Humières, P. Lallemand, and U. Frisch, Europhys. Lett., 2, p.291, 1986.

[3] C. M. Teixeira, "Continuum Limit of Lattice Gas Fluid Dynamics", Doctor thesis of MIT, 1992.

[4] H. Matsuoka and N. Kikuchi, "Controlling Effects on the viscosity of simulated fluid by using the method of multi-collision of two particles" (in Japanese), SENAC, Vol.44, No.3, pp.21-28, 2011.7.

[5] Sadatoshi Taneda, "*Gazou kara manabu ryuutairikigaku*" (in Japanese), Asakura-shoten, p.45 and p.47.

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

著者

松岡　　浩　　東北大学

菊池　範子　　株式会社サイエンス・サービス

渡辺　　正　　福井大学

横川三津男　　神戸大学

峯尾　真一　　高度情報科学技術研究機構

板倉　憲一　　海洋研究開発機構　地球シミュレータセンター

岩谷　幸雄　　東北大学

藤縄　幸雄　　藤縄地震研究所

折田　義彦　　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体衝突法”とよぶ粘性制御法を用いて、高レイノルズ数状態にある流体流れのシミュレーションを実現することを目指す。本手法の適用にあたっては、各格子点においてランダムに選びだされた２粒子衝突を多数回繰り返し、その後に粒子の並進移動を行う。さらに、2012 年度の研究では、３速さ 54 速度モデルを導入した。本モデルは、テシャラが 1992 年の博士論文で公表したものであり、粒子の平均自由行程を１格子点間隔未満にすることができる。本手法を用いることで、格子点数を増やさずにある程度高いレイノルズ数状態にある流体流れをシミュレーションすることができる。また、本手法の特徴は、“大規模並列ビット演算を行うこと”及び“連続の式とナビエ・ストークス方程式に従うマクロな挙動を正しく再現できること”である。

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