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 The Earth Simulator Center, Japan Agency for Marine-Earth Science and Technology

*6 Fujinawa Earthquake Research & Development

*7 Japan Marine Science Foundation

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 92.8% as the calculation-efficiency by using 576 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

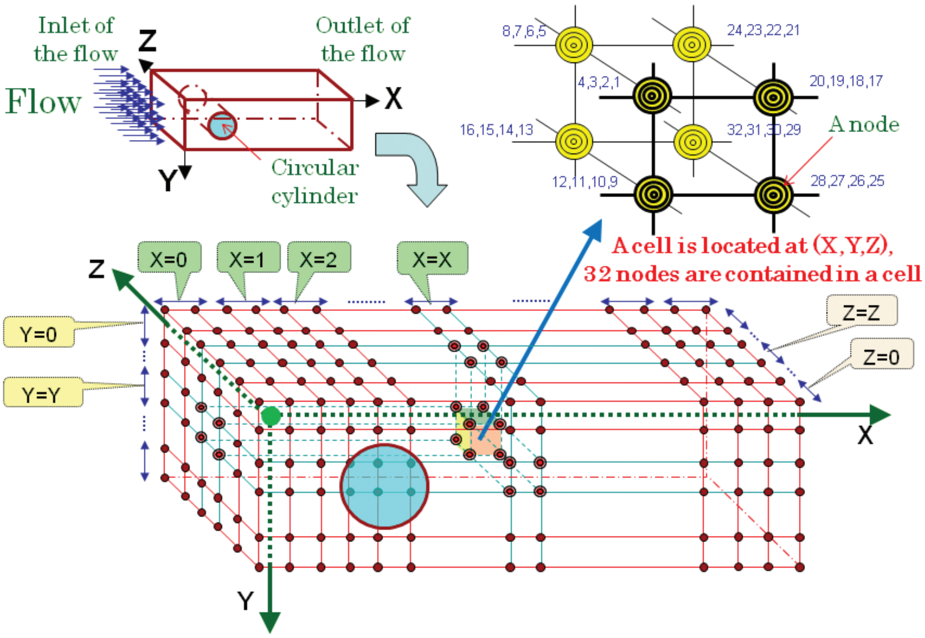


Fig. 1 Cells and nodes for simulating the three dimensional vortices shedding from a circular cylinder.

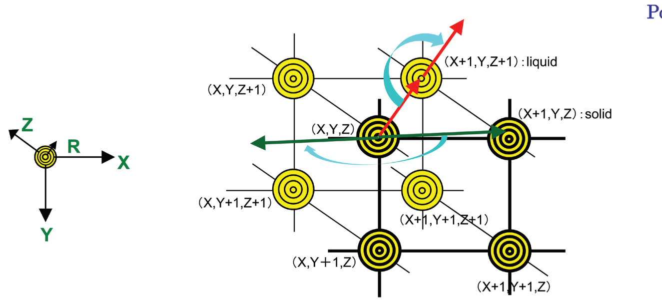
have been found, such that two- and three-dimensional fluid dynamics are recovered in the macroscopic limit. The class of cellular automata used for the simulation of fluid dynamics is called "lattice gas models", and many lattice gas models have been proposed.

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'Humiéres, 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.



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.

Possible 24 velocities of particles

	(ΔX, ΔY, ΔZ, ΔR)
1	(0 0 −1 −1)
2	(0 0 −1 1)
3	(0 −1 0 −1)
4	(0 −1 0 1)
5	(0 −1 −1 0)
6	(0 −1 1 0)
7	(−1 0 0 −1)
8	(−1 0 0 1)
9	(−1 0 −1 0)
10	(−1 0 1 0)
11	(−1 −1 0 0)
12	(−1 1 0 0)
13	(1 −1 0 0)
14	(1 1 0 0)
15	(1 0 −1 0)
16	(1 0 1 0)
17	(1 0 0 −1)
18	(1 0 0 1)
19	(0 1 −1 0)
20	(0 1 1 0)
21	(0 1 0 −1)
22	(0 1 0 1)
23	(0 0 1 −1)
24	(0 0 1 1)

Fig. 2 Propagation rules from node to node.

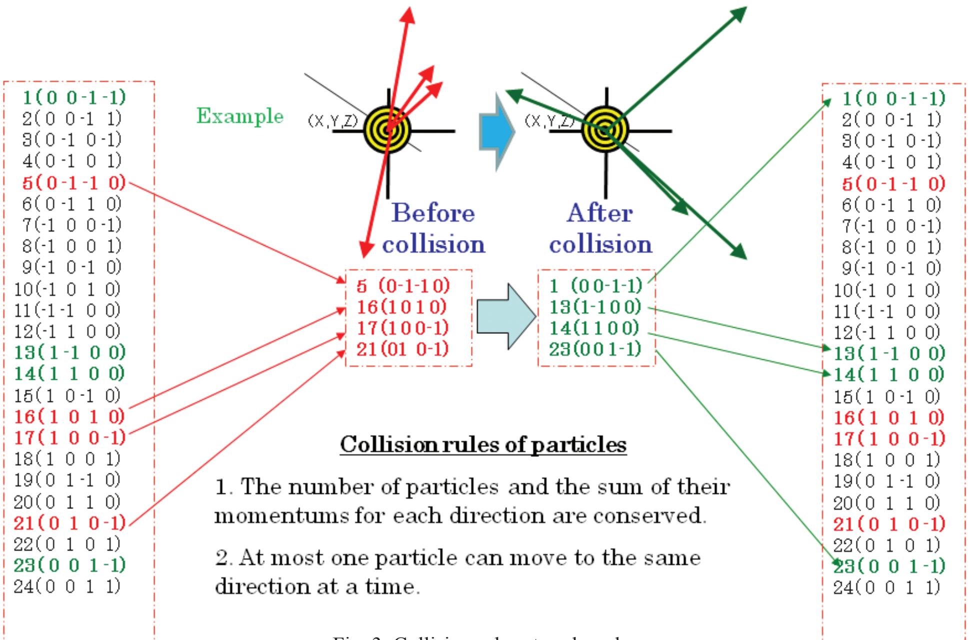


Fig. 3 Collision rules at each node.

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, (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), 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.

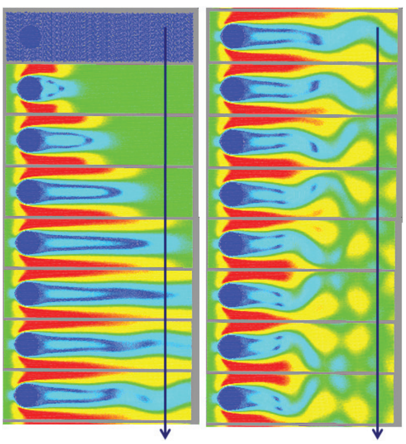


Fig. 4 A transient simulation result of the flow past a circular cylinder of infinite lenngth.

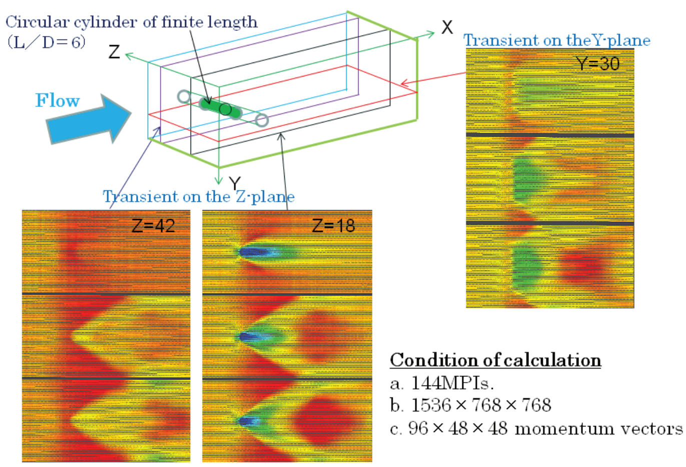


Fig. 5 Preliminary simulation of the flow past a circular cylinder of finite length.

3.3 Results of the study in FY 2010

Regarding the improvement of the calculation-efficiency of "massively parallel bit-wise calculations", we obtained the value of 92.8% by using 576 vector CPUs of Earth Simulator. We regard that the value is a good performance.

As for the three-dimensional simulation, we, at first, 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. We, secondly, tried to simulate the vortices shedding from a circular cylinder of finite length. One of the results is shown in Fig.5. There seems to be indications similar to the calculated result shown by Inoue and Sakuragi[4]. Further study is need in FY 2011.

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'Humiéres, B. Hasslacher, P. Lallemand, Y. Pomeau, and J-P. Rivet, "Lattice Gas Hydrodynamics in Two and Three Dimensions", Complex Systems, 1(1987), pp.649-707.

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

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

[4] Osamu Inoue and Akira Sakuragi, "Vortex shedding from a circular cylinder of finite length at low Reynolds numbers", PHYSICS OF FLUIDS, 20(2008), 033601.

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]，峯尾　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めざす。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。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 576 個のベクトル CPU を用いて、並列化効率 92.8% という値を得た。

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