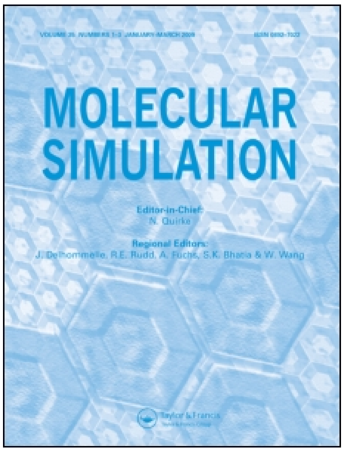


Molecular Simulation
Publication details, including instructions for authors and subscription information:
http://www.informaworld.com/smpp/title~content=t713644482

Computer Simulation of Two-Dimensional Continuum Flows by the Direct Simulation Monte Carlo Method

Hideo Kaburaki[a]; Mitsuo Yokokawa[a]
[a] Computing and Information Systems Center, Japan Atomic Energy Research Institute, Naka-gun, Ibaraki, Japan

To cite this Article Kaburaki, Hideo and Yokokawa, Mitsuo(1994) 'Computer Simulation of Two-Dimensional Continuum Flows by the Direct Simulation Monte Carlo Method', Molecular Simulation, 12: 3, 441 — 444
To link to this Article: DOI: 10.1080/08927029408023052
URL: http://dx.doi.org/10.1080/08927029408023052

Molecular Simulation, 1994, Vol. 12(3-6), pp. 441-444
Reprints available directly from the publisher
Photocopying permitted by license only

Printed in the United States of America

Note

COMPUTER SIMULATION OF TWO-DIMENSIONAL CONTINUUM FLOWS BY THE DIRECT SIMULATION MONTE CARLO METHOD

HIDEO KABURAKI and MITSUO YOKOKAWA

Computing and Information Systems Center, Japan Atomic Energy Research Institute, Tokai-mura, Naka-gun, Ibaraki 319-11, Japan

(Received March 1993, accepted July 1993)

KEY WORDS: Direct simulation Monte Carlo method, lid-driven cavity flow, continuum flow

Computer simulations using particles are an attractive method to extract microscopic information of flow phenomena [1]. The molecular dynamics (MD) method, in which Newton's equations of motions are integrated, gives the temporal development of the system. In the MD simulation of fluid flows, the computational region is limited to atomistic scales [2]. On the other hand, the direct simulation Monte Carlo (DSMC) method, in which collisions of particles are made on a probabilistic basis, has a potential of treating a realistic system with a macroscopic scale length retaining the atomistic details. The DSMC method provides an efficient way to integrate the Boltzmann equation from the rarefied gas to the near-continuum region. Bird clarified the validity of the DSMC method in the near-continuum flow region [3]. However, the DSMC method has not been applied to the continuum region and compared with the continuum hydrodynamics.

The present paper studies the applicability of the DSMC method to the continuum flows whose motions are described by the Navier-Stokes equation. The lid-driven cavity flow has been simulated and the results are compared with those by the continuum hydrodynamics. The usual procedure for the DSMC algorithm is adopted [4]. The conventional Bird's Time Counter (TC) method is used in the collision process. Bird [5] also proposed the No Time Counter (NTC) method to alleviate difficulties of the TC method encountered in hypersonic problems. However, for the incompressible flows considered here, the two methods make no difference. In the simulation, collision subcells are generated in a sampling cell by monitoring the local number density, so that collisions take place in collision subcells whose length is less than the local mean free path of the field [6]. With collision subcells, the requirement on the length of the collision cell is strictly met throughout the calculation.

The test case is the two dimensional lid-driven cavity flow in the continuum region ($Kn = 2.96 \times 10^{-3}$) and at the Reynolds number Re of 100. This flow is selected

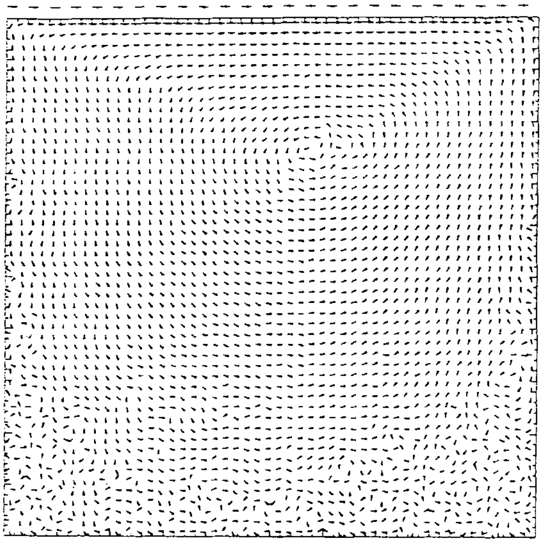
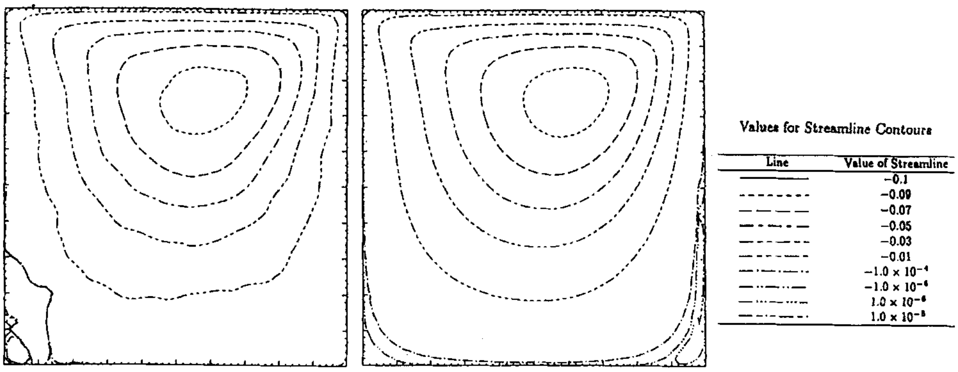


Figure 1 The averaged velocity vectors with 300 samplings.

because a reliable solution of the Navier-Stokes equation by using the finite difference method is available in this region. For the simulation of the continuum hydrodynamics, the finite difference method (the Marker and Cell method) [7] is employed.

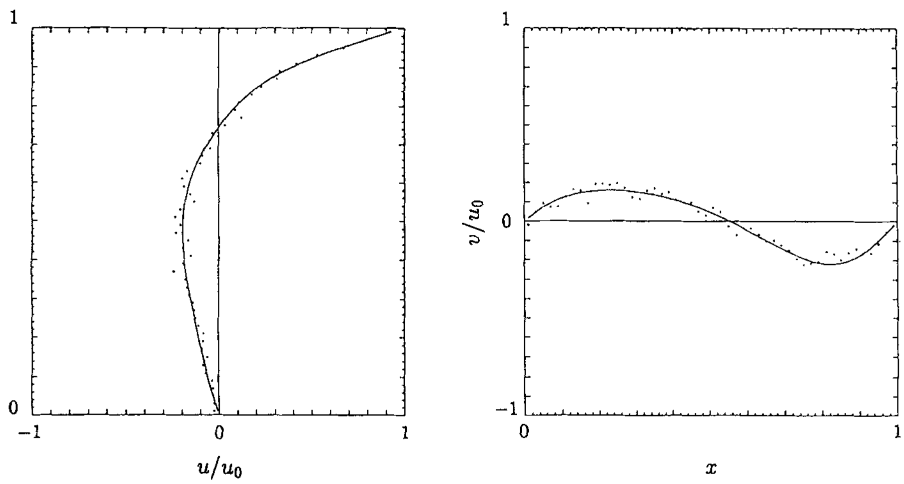
In the simulation by the DSMC method, the computational region is divided into $50 \times 50 \times 1$ sampling cells where each sampling cell consists of nearly 50 collision subcells. The number of particles used in the simulation is 1.75×10^6, which means nearly 15 particles in a collision subcell. The upper lid moves at a constant speed and its Mach number is assumed to be 0.2. The time step is 0.9 times the mean free time of the gas. The sampling is taken at an interval of 2 to 3 mean free times after the flow comes to a steady state. The total number of samplings is 300. This calculation is performed on the Fujitsu VP2600/10 and the CPU time and memories are approximately 200 hours and 200MB, respectively.

The averaged velocity field is shown in Fig. 1 and a primary vortex is seen to be clearly formed. Figures 2 (a) and (b) show numerical results of the streamlines by the DSMC method and the finite difference method, respectively. The contour levels are denoted in Fig. 2. It is seen that the stream lines by the DSMC method are in good agreement with those by the finite difference method in most of the region of the cavity. In the bottom region of the cavity where the stream function has small values in magnitude, the DSMC method cannot reproduce correctly the velocity field due to statistical fluctuations. Figures 3 (a) and (b) show the velocity profiles along the vertical and horizontal lines through the center of the cavity. The line in the figure is the result by the finite difference method and the dots are the results by the DSMC method. The agreement between the two results is satisfactory.



(a) DSMC method (b) Finite difference method

Figure 2 The streamlines by (a) the DSMC method and (b) the finite difference method.



(a) (b)

Figure 3 The velocity profiles along (a) the vertical and (b) the horizontal line. The velocities are normalized by the velocity of the lid (u_0).

With the right collision cells and proper number of particles, it is shown that the DSMC method is able to simulate correctly the gas flows in the region where the Navier-Stokes equation holds.

References

[1] A.L. Garcia, "Hydrodynamic fluctuations and the direct simulation Monte Carlo method" in *Microscopic Simulations of Complex Flows*, M. Mareschal ed. Plenum Press, New York and London, 1990, p. 177.

[2] D.C. Rapaport, "Microscale hydrodynamics: Discrete-particle simulation of evolving flow patterns", *Phys. Rev.*, A36, 3288 (1987).

[3] G.A. Bird, "Direct simulation of high-vorticity gas flows", *Phys. Fluids*, **30**, 364 (1987).

[4] G.A. Bird, *Molecular Gas Dynamics*, Oxford University Press (1976).

[5] G.A. Bird, *Rarefied Gas Dynamics*, Progress in Astronautics and Aeronautics Series, Vol. 118 (AIAA, Washington DC), 1989, p. 211.

[6] M. Yokokawa, K. Watanabe, H. Yamamoto, M. Fujisaki and H. Kaburaki, "Parallel processing for the direct simulation Monte Carlo method", Computational Fluid Dynamics Journal 1(3), 337 (1992).

[7] F.H. Harlow and J.E. Welch, "Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface", *Phys. Fluids*, **8**, 2182 (1965).