Simulation of a Flow around a Bluff Body Using Cartesian Coordinates

Kunio Kuwahara

Institute of Computational Fluid Dynamics, Meguro-ku, Tokyo 152-0011, Japan

Satoko Komurasaki

Department of Mathematics, College of Science and Technology, Nihon University, Chiyoda-ku, Tokyo 101-8308, Japan

Angel Bethancourt

Institute of Computational Fluid Dynamics, Meguro-ku, Tokyo 152-0011, Japan

This paper introduces a computational technique to compensate for the added numerical diffusion that is generated when uniform Cartesian coordinates are used to describe the flow around bluff bodies. Because of the staircase-like representation of the surface object, it was found that the added surface "roughness" causes larger than expected separation region for some test cases (flow around a sphere, and flow around a cylindrical body). In order to control the velocity profile in the boundary layer, a new parameter *blvr* (boundary layer velocity ratio) is defined, and it is used to set the negative value of the viscosity along the surface. Extensive visualizations of flow past bluff bodies are performed using the present technique. Numerical solutions of the governing Navier-Stokes equations are carried out in a uniform Cartesian coordinates using a multi-directional finite difference scheme with a third-order upwinding. No explicit turbulence model is incorporated into the model, and the dependence of the solution on the *blvr* parameter is investigated.

Nomenclature

u Velocity vector

p Pressure difference from the base pressure

t Time

δt Time increment in numerical time integration

Re Reynolds number based on chord length

$blvr$ Velocity ratio in the boundary layer

Superscript

n Time step

I. Introduction

Many simulations of a flow past streamlined body have been carried out, mostly using a finite-difference method in a body-fitted coordinate system.[1] Simulations of bluff bodies are less frequent because of difficulties in solver implementation and grid generation. It is often acknowledged, that grid generation is one of the most difficult and manpower consuming parts when dealing with body-fitted coordinates. On the other hand, many important applications involve flow around a bluff body, i.e. flow around a car. The simplest way to avoid all these complications would be to employ a Cartesian coordinate system in which the body is represented by creating a masking data (voxel) on the grid coordinates.[2] However, this approach results in lack of resolution near the boundary of the object. If the object is visualized, it looks like a staircase structure. By using multi-directional finite differences,[3,4] a smoother representation of the staircase boundary is achieved, but the resolution problem is not completely solved. In problems involving separation depending on the resolution of the boundary layer, simulations predict larger than expected areas of separation. This is due to the numerical diffusion caused by the roughness on the boundary. To resolve the behavior of the flow along the staircase boundary, we introduce a negative viscosity on the surface of the body to compensate for the numerical diffusion. In the present paper, several examples using this technique are presented.

II. Computational Method

The governing equations are the incompressible Navier-Stokes equations. In Cartesian coordinates system, they can be written as it follows,

$$\frac{\partial u_1}{\partial x_1} + \frac{\partial u_2}{\partial x_2} + \frac{\partial u_3}{\partial x_3} = 0, \tag{1}$$
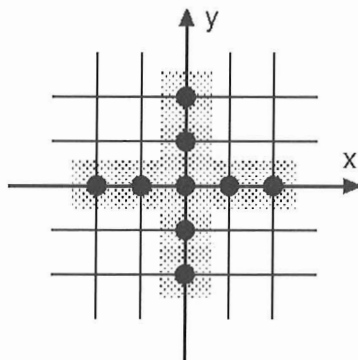
$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left\{\frac{1}{Re}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right\}. \tag{2}$$

For high-Reynolds number flow, time-dependent computations are required owing to the strong unsteadiness. Presently, a finite-difference method is employed to discretize the basic equations and they are solved using the projection method (Chorin,[5] and Takami and Kuwahara[6]). The pressure field is obtained by solving the following Poisson's equation:
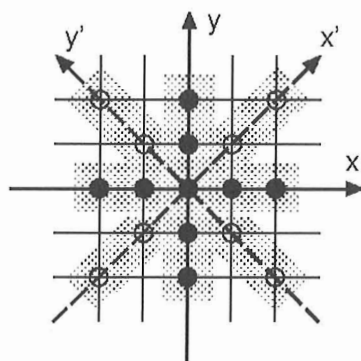
$$\Delta p = -div(u \cdot grad u) + \frac{D^n}{\delta t}, \quad D = div u \tag{3}$$

where n is the time step and δt is the time increment. D^{n+1} is assumed to be zero, but D^n is retained as a corrective term.
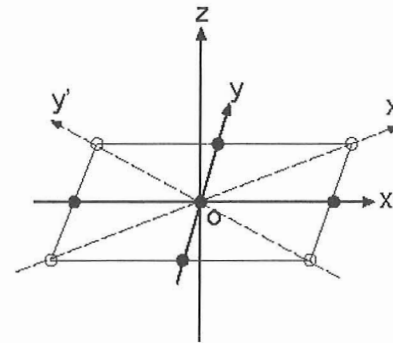
In the present paper, a multi-directional finite difference method is implemented when discretizing the governing equations. In case of 2-dimensional computations, when structured grid points are given, the black points in Fig. 1(a) are usually used to approximate the derivatives at the central point (system A). If we introduce another 45° rotated local grid system, the white points in Fig. 1(b), can be used to approximate the derivative at the central point (system B). In order to improve the derivative value at the central point, the values of both systems are combined. If a ratio A: B=2:1 is adopted, the resulting finite difference scheme for the Laplacian coincides with the well-known 9 point formula with fourth-order accuracy. This method improves the rotational invariance of the coordinate system, and then those cases where flow direction is not parallel to the grid location are better simulated. In 3 dimensions, three different grid systems are used. Each grid system is obtained by rotating a perpendicular plane 45° with respect to each coordinate axis. One of such systems is shown in Fig. 1(c).

(c) System Z: $x' - y' - z$.

Figure 1. System for multi-directional scheme

(a) System A.

(b) System B.

Space derivatives are discretized using a three-point central difference approximation with exception of the convective terms. For the convective terms, a third-order upwind scheme is used to stabilize the computation (Kawamura[7]). It has been found to be the most suitable for high-Reynolds number flow computations. The second-order Crank-Nicolson implicit scheme is used for time integration. The equations are iteratively solved at each time step by SOR method. A multi-grid method is utilized to solve the Poisson's equation.

Negative viscosity

In the Cartesian coordinate system, the body is represented as a set of voxels at the grid points with their values set on/off (a binary operator) to indicate its presence. As stated before, the surface of the body resembles a staircase. After computations are carried out, larger than expected regions of separations are observed. This is due to the numerical diffusion caused by the surface roughness. Therefore, a special treatment of the boundary conditions is needed in order to properly simulate these types of flows. The present technique introduces a negative value of the viscosity on the surface of the body to compensate for the numerical diffusion. At high-Reynolds number, turbulence in the free space is simulated without using an explicit turbulence model in these computations. Viscous effects are limited only within the boundary layer. Therefore, a simple model is made for the boundary layer to account for the viscous effect.

It is important to point out that this negative viscosity has no definitive physical meaning, and how to determine its value represent a big challenge. If the absolute value is large enough, the flow near the boundary accelerates and separation is reduced. On the other hand, if the absolute value is small, the separation region becomes larger. Therefore, research in a proper way to determine this value is undertaken. The velocity profile in the boundary layer is closely related to the negative viscosity. The present paper defines a parameter *blvr* (boundary layer velocity ratio) that it is used to determine the proper value of the negative viscosity (See Figure 2).

The parameter $blvr$ is the ratio of the averaged velocity between the two points nearest to the surface ($blvr = v1/v2$). If $blvr$ is 0.5, the local flow Reynolds number is 0.0. On the other hand, if it is 1.0, a free-slip condition is imposed $blvr=v1/v2$ on the surface. Therefore, the value of the $blvr$ parameter, as described in Figure 2, shall fall between 0.5 ～1.0. Presently, the value of the negative viscosity ratio ($blvr$) is a function of this parameter.
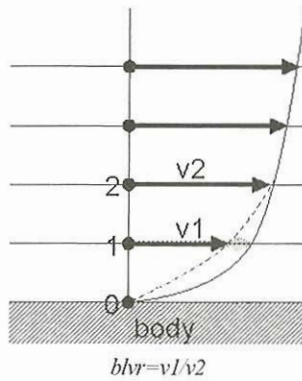

$$blvr=v1/v2$$

Figure 2.　Boundary layer velocity ratio ($blvr$)

III. Computational Results

Examples of 3-dimensional simulation of flow around bluff body with using the negative viscosity are visualized.

Figure 3 shows computational grid and a body represented by a set of voxels for simulation of a flow around a sphere. Figure 4 explains the effect of the negative viscosity, and $blvr$ in Fig. 4 (a)～(c) are 0.60, 0.70 and 0.75 respectively.

As other examples of simulation, flows past a rectangular cylinder with 33% rounded corners at Re = 500 and 1, 000, 000 are visualized in Figs. 5 and 6, respectively. Both flows are simulated with $blvr$ =0.70. From these figures, it is shown clearly that both low and high Reynolds number flows are well captured by using the present technique.
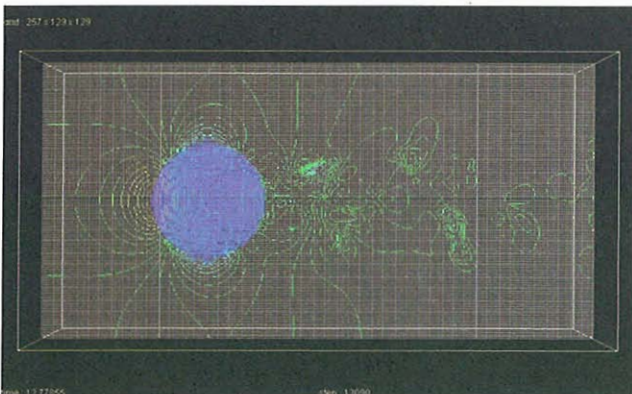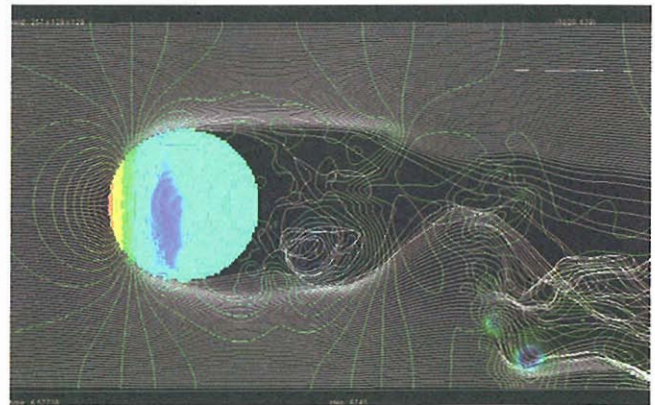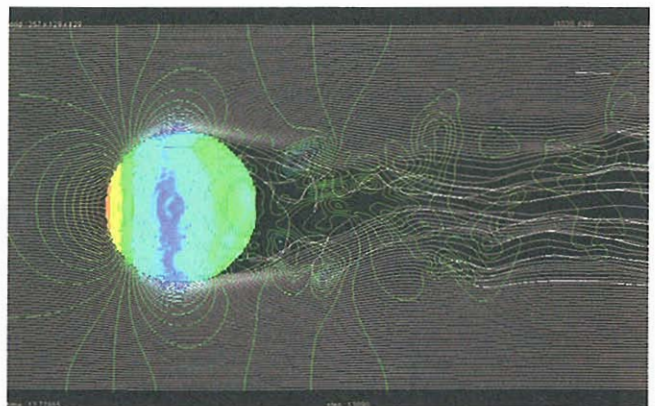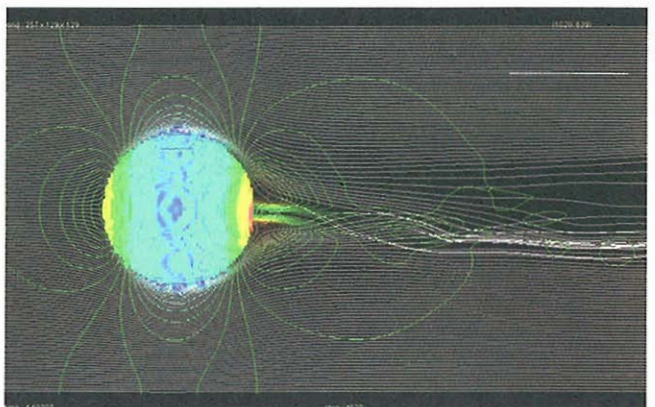

Figure 3.　Computational grid for simulation of flow around a sphere
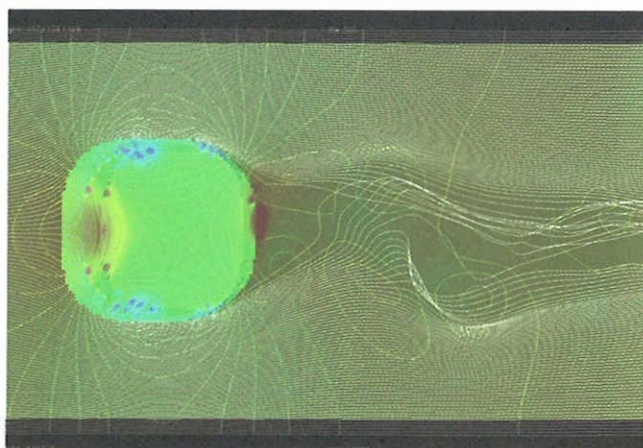

(a) $blvr$ =0.60 (larger separation areas).

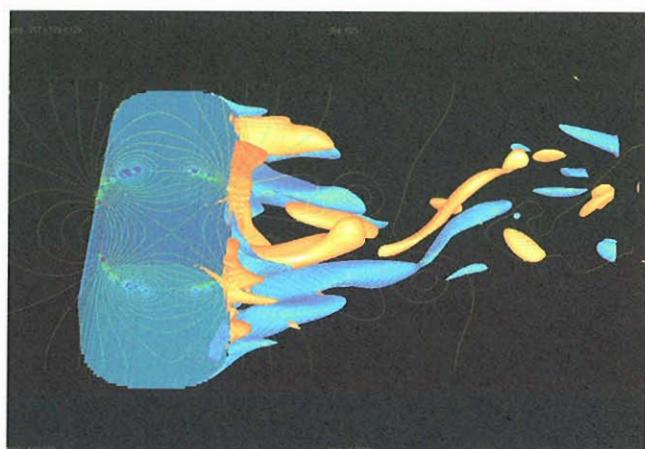(b) $blvr$ =0.70 (occurrence of drag crisis: standard case).

(c) $blvr$ =0.75 (too large $blvr$).

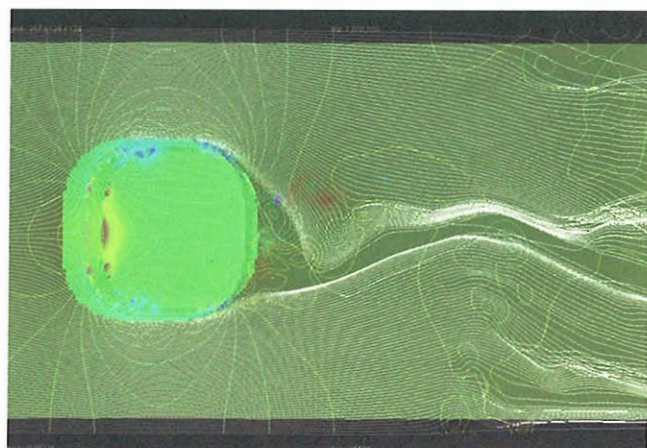Figure 4.　Flow around a sphere in each $blvr$; pressure field and stream lines.
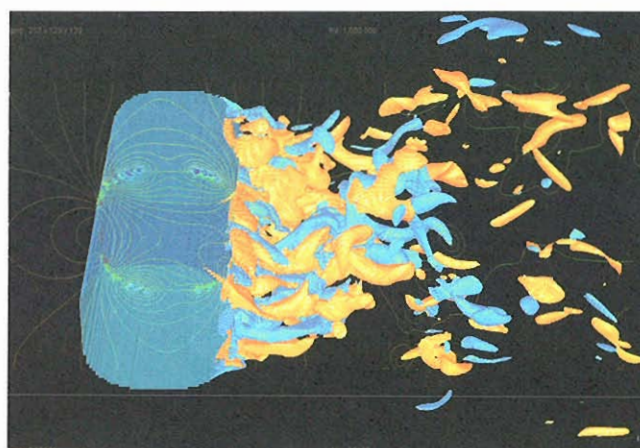
(a) Stream lines and pressure shading.

(b) Longitudinal component of vorticity.

Figure 6. Flow around a cylindrical body at Re = 1, 000, 000.

(b) Longitudinal component of vorticity.

Figure 5. Flow around a cylindrical body at Re = 500.

(a) Stream lines and pressure shading.

IV. Conclusion

Three dimensional flows around bluff bodies were simulated in Cartesian coordinate system. In this system, the separation was effectively reduced by using the negative viscosity. Also, flow fields were clearly captured consistently.

Extension to compressible fluid flow is easy and straightforward.

References

1) Komurasaki, S. and Kuwahara, K., 2004, "Implicit Large Eddy Simulation of a Subsonic Flow around NACA0012 Airfoil," AIAA Paper 2004-0594

2) Bethancourt, A, Kuwahara, K. and Komurasaki, S., 2003, "Grid Generation and Unsteady Flow Simulation around Bluff Bodies," AIAA Paper 2003-1129

3) Kuwahara, K., 1999, "Unsteady Flow Simulation and Its Visualization," AIAA Paper 99-3405

4) Kuwahara, K. and Komurasaki, S., 2002, "Simulation of High Reynolds Number Flows Using Multidirectional Upwind Scheme," AIAA Paper 2002-0133

5) Chorin, A., J., 1968, Math. Comp. 22 pp. 745

6) Takami, H. and Kuwahara, K., 1974,"Numerical Study of Three-Dimensional Flow within a Cubic Cavity," J. Phys. Soc. Japan, Vol. 37, No. 6

7) Kawamura, T. and Kuwahara, K., 1984,"Computation of high Reynolds number flow around a circular cylinder with surface roughness," AIAA Paper 84-0340.