Numerical Scheme to Resolve the Interaction between Solid Particles and Fluid Turbulence*

Satoshi TAKIGUCHI**, Takeo KAJISHIMA** and Yutaka MIYAKE**

We developed an efficient numerical scheme to resolve the flow around a solid particle that is several times larger than the spacing of computational grid. The method was examined by applying it to the three-dimensional flow past a sphere in a uniform stream. For the case that the grid resolution is finer than 1/8 of particle diameter, our results agreed well with experiment up to Reynolds number 600. It evaluated exactly the momentum transfer due to pressure at the particle surface. The numerical error in the energy budget was less than 6% of the dissipation rate. Our method therefore has enough accuracy and efficiency to investigate the interaction between particle-induced eddies and fluid turbulence in particle-laden turbulent flows.

Key Words: Mulitphase Flow, Turbulent Flow, Finite Difference Method, Wake, Direct Numerical Simulation, Particle-Laden Turbulence, Gas-Solid Two-Phase Flow

1. Introduction

In this paper, we propose a numerical scheme for two-phase flows including dispersed phase such as solid particles. The aim of our study is to improve turbulence models suited for multiphase flows by the aid of the direct numerical simulation (DNS). To this end, we developed a method to capture the flow around each particle by a fixed grid system for fluid turbulence. Typical characteristics in our approach are the sufficient resolution for flows around each particle, the accurate evaluation of momentum as well as energy exchange between fluid and particles, and the efficiency in tracking many particles. We guess this method could provide an efficient tool for wide variety of flow fields; for example, flows around bodies of complicated geometry, flows through many obstacles, flow around a moving bodies with fluid-solid interaction, and so on.

To parameterize the turbulence modulation in multiphase flows, we must take account of the energy production and dissipation due to flow around particles. Gore and Crowe analyzed many experimental data and concluded the ratio between particle diameter and integral scale of turbulence is a governing factor of turbulence modulation. That is, the ratio 0.1 is a threshold of increase/decrease in turbulence intensity. Elghobashi, on the other hand, proposed a mapping by the volumetric fraction and time-scale ratio between particle motion and fluid turbulence. In spite of these intensive investigations, the mechanism of turbulence modulation has not been explained completely. Thus the general description of multiphase turbulence is not available yet. Therefore, we should clarify the turbulence structure concerning to the modification in turbulence energy and Reynolds stress through the observations of particle motion in turbulence and the flows around particles.

For the first step of our research, we propose a finite-difference method to resolve the particle-induced eddies in the present paper. We will apply this method to the DNS of particle-laden turbulence. The
present approach has a limitation in scale range considering practical calculations of particle-laden flows. Our contribution to practical computations may be through the development and evaluation of turbulence models by the DNS.

Recent numerical investigations\(^{(3)-(6)}\) have reproduced complex behaviors of multiphase fluid by tracking the motion of every particle. In these studies, the fluid phase was simulated directly by the computation of Navier–Stokes equation of motion just like the DNS of single-phase turbulence. Particles, however, were tracked by the point-force model\(^{(7)-(9)}\). These equations of particle motion are basically for the small sphere in a uniform stream and they are composed of drag force term due to fluid viscosity and pressure, virtual mass term, Basset's memory term, buoyancy/gravity term, Saffman's lift force term and so on. It is no more than the mixing of modeled particles into the DNS of single-phase flows. Kenning and Crowe\(^{(10)}\) pointed out that such an approach is not sufficient for basic research of turbulence modulation due to particles. Problems in using the point-source model may be as follows. Firstly, the accuracy of equation of motion in the actual turbulence has not yet been confirmed although some researchers improved the point-source models\(^{(11)-(12)}\). They could access the performance of system equation but could not evaluate each term in the equation. Secondly, the point-force model does not take the particle-induced eddies into account. It results in a significant uncertainty in elucidating the mechanism of turbulence modulation, which is one of the most important factors in parameterization of multiphase turbulence. Thirdly, it is difficult to apply the point force model to the non-spherical particles.

Some researchers tried a pure DNS which did not contain any empirical and/or theoretical models concerning to the particle motion. Hus\(^{(13)}\) filled the fluid domain by the unstructured grid system that moves according to the particle motion. He integrated the stress on the particle surface to obtain the fluid force on the particle. It was the two-dimensional simulation. We guess the fully three-dimensional simulation is difficult by such an approach due to crucial requirement for computer resources. Pan and Banerjee\(^{(14)}\) did not use a no-slip boundary condition or a non-permeable condition exactly at the interface between solid and fluid. They added the external force at the grid points occupied by particles to reproduce a flow around particles.

A DNS method to resolve the interaction between turbulence eddies and solid particles should be designed taking the above-mentioned characteristics of multi-phase turbulence into account. Settling boundary-fitted grids for many particles is not efficient for such a purpose because of a significant increase in computer capacity. In the present study, we developed a method to simulate a flow field including particles that move with vortex shedding. The numerical example shown in this paper is the flow around a fixed sphere in a uniform stream. We pay our attention not only on the numerical accuracy for fluid force on the particle but also the evaluation of energy budget that is necessary for turbulence modeling.

2. Numerical Method

The particle-laden flow field is simulated by the finite-difference method using a fixed grid system, mostly in the Cartesian coordinate in our method. We assume the scale of particles in the flow field is several times larger than the computational grid but there is no special restriction in their shape.

2.1 Basic equations

We consider a gas-solid or liquid-solid two-phase flow assuming the fluid phase to be incompressible and Newtonian and the particles to be rigid. The governing equations for fluid flow are the mass continuity equation

$$\nabla \cdot \mathbf{u} = 0$$

(1)

and the Navier–Stokes equation of motion

$$\frac{D \mathbf{u}}{Dt} = \nabla \cdot \mathbf{\tau} + \rho \mathbf{g},$$

(2)

where \( \mathbf{u} \) denotes the fluid velocity, \( \mathbf{\tau} \) the stress tensor

$$\mathbf{\tau} = -p \mathbf{I} + \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)$$

(3)

\( p \) the static pressure, \( \rho \) the fluid density, \( \mu \) the fluid viscosity. The second term in the right-hand side of Eq. (2) represents the external force such as gravity/buoyancy and it does not include a modeled force term due to particles.

Solid particles, on the other hand, are tracked by the equations of momentum

$$\frac{dm}{dt} = \int_{S_p} \mathbf{r} \cdot dS + \mathbf{G},$$

(4)

and angular momentum

$$\frac{dL}{dt} = \int_{S_p} \mathbf{r} \times (\mathbf{r} \cdot \mathbf{n}) dS + \mathbf{N},$$

(5)

in the Lagrangian frame of reference, where \( \mathbf{v} \) is the particle velocity, \( \mathbf{\omega} \) the angular velocity, \( m_p \) the mass of particle. The inertia tensor \( I_p \) is represented by \( I_p = (2/5)a^2m_p I \) for a solid sphere of a radius \( a \). In the right hand sides of Eqs. (4) and (5), \( \mathbf{G} \) and \( \mathbf{N} \) are the external force and moment on the particle, respectively, but they do not include the force or moment due to fluid flow. The surface integrals in Eqs. (4) and (5) give the influences of the fluid flow, where \( S_p \) is the particle surface, \( \mathbf{n} \) the normal unit vector in the outside direction at the surface, \( \mathbf{r} \) the relative distance
vector from the center of moment to the surface. Note that the basic equations for particle motion do not include any empirical models for drag force, lift force, memory term or additional terms.

2.2 DNS of fluid turbulence

The basic equations are spatially discretized by a staggered arrangement on the Grid system in the Cartesian coordinate \((x_1=x, x_2=y, x_3=z)\). The grid spacing in each direction, \(h_x, h_y, h_z\), is uniform in this case but it is not a condition of our scheme. The mass continuity equation (1) in a finite-difference form at the center \((x_i = Ih_x, y_i = Ih_y, z_i = Kh_z)\) of the cell indexed as \((I, J, K)\) is

\[
\frac{\partial \rho_t}{\partial t} + \frac{\partial}{\partial x_i} (\rho_t u_i) = 0.
\]

(6)

At the center of cell interface, the normal-to-interface component in the equation of motion is discretized as

\[
\frac{\partial \rho_t u_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho_t u_i u_j) = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + g_i,
\]

(7)

where \(\nu = \mu / \rho\) is the kinematic viscosity of the fluid. The convective-interpolation method\(^{(19)}\) used in the second term of the left-hand side has consistency and energy conservation properties.

The operators, \(\delta_i\) and \(\overrightarrow{\cdot}\), represent the finite-difference and the interpolation, respectively. They use symmetrical stencils \(\pm(m - \frac{1}{2})\), where, \(m = 1, 2, \ldots, M/2\), without any upstream bias, resulting in the \(M\)-th order accuracy in space. Here in this work, we choose the fourth order schemes. The finite-difference and interpolation in the \(x\) direction at the location \(I\), for example, are represented as

\[
\delta_i f_i = \frac{1}{24h_x} (f_{i-3} + 2f_{i-1} - 2f_{i+1} - 2f_{i+3}),
\]

(8)

\[
\overrightarrow{f^*} = \frac{1}{16} (-f_{i-3} + f_{i-1} + 9f_{i-1} + 9f_{i+1} - f_{i+3}),
\]

(9)

where subscripts for other directions, \(J\) and \(K\), are omitted for simplicity.

We have applied this fourth order scheme to DNS of a decaying isotropic turbulence\(^{(18)}\) and a turbulent flow in a plane channel\(^{(17)}\) and confirmed the reliability by comparing our results with experimental and other DNS results.

2.3 Numerical scheme for flow around particles

In the region close to a particle as shown in Fig. 1, we use the third order interpolation in the Cubic-Interpolated Pseudo-Particle (CIP) method\(^{(18)}\). However, we reduce the calculation of transport equation for velocity gradient to a first-order upstream method. The reason of this is to save the computer resources considering a large number of particles in the simulation of our purpose. We do not think this simplification could affect the entire accuracy as shown later. The region for such a simplified-CIP scheme is where one or more stencils in finite-difference or interpolation equation are inside the solid body. As for Eqs. (8) and (9), it is a cubic region that is larger than a particle by two mesh spacing in every direction.

A diagonalized form\(^{(19)}\) of the CIP method for an equation \(\frac{\partial}{\partial t} \rho_t + \rho_t \overrightarrow{\partial x} = \overrightarrow{g}\) is represented as

\[
f^* = \frac{13}{32} (f_{i-1} + 2f_{i+1}) + \frac{5h_x}{192} (f_{i+1} - f_{i-1})
\]

\[
+ \frac{1}{h_x} (\Delta F_{i-1/2} - \Delta F_{i+1/2}),
\]

(10)

which is shown one-dimensionally for simplicity. In Eq.(10), \(f^* = f^{(n)} + \Delta f^{(n)}\) and \(f^* = f^{(n)} - \Delta f^{(n)}\) are explicit predictions for \(f\) and \(f^*\) by the first order Euler method. We evaluated \(f = f^{(n)} (x - \Delta x)\) and \(\Delta F\) by the third order interpolation same as the CIP method, where \(\bar{f}\) is an approximation for \(f^{(n+1)}(x)\), and \(\Delta F\) the flux, see the references\(^{(18,19)}\) for detail. On the contrary, we do not calculate the equation for \(f^*\) but give \(f^*\) by the first order upstream scheme as

\[
f^* = \begin{cases} (f_{i-1} + f_{i+1})/h_x, \ &u_i > 0 \\ (f_{i-1} + f_{i+1})/h_x, \ &u_i < 0 \end{cases}
\]

(11)

It is slightly different from an upstream method derived at the early stage of CIP method\(^{(18)}\) while those could have similar effect. Our purpose for simplification, Eq.(11) in particular, is not to introduce a numerical viscosity for stabilization but to save computational time and capacity. The first order upstream method is preferred since this is used only in the region very close to the solid particles where the downstream side of finite-difference stencil enters the solid phase. Note that this treatment is limited in the vicinity of the solid particles.

When the finite-difference stencil for the viscous term in the equation of motion enters the solid body, we move it to the location at the boundary to enforce the no-slip boundary condition. The finite-difference stencils for the viscous term therefore become non-uniform.
2.4 Evaluation of fluid force on particles

While the numerical integral of Eqs.(4) and (5) might be possible even by the relatively poor resolution of mesh, we need some improvement for higher accuracy.

When the slip velocity between a particle and fluid become larger, the pressure drag is getting more dominant among fluid forces on the particle. The accuracy for this term is of particular importance. In addition, we wish to eliminate an error in momentum exchange through the interface of two phases. A possible source of this error is the difference between the surface integral of pressure in Eq.(4) for particle motion and the volumetric integral of the pressure gradient in Eq.(2) for fluid flow. We therefore replaced the surface integral in Eq.(4) by the volumetric integral as

\[ F_{p} = -\int_{S_{p}} p dS = -\int_{S_{p}} \nabla p dV \]  

using Gauss divergence theorem. As the same pressure gradient in fluid, \( \nabla p \), is used for both the equations of motions of particles as well as fluid, error in the momentum budget is completely removed. As for the equation of angular momentum, the surface integral is replaced by the volumetric integral as

\[ T_{v} = -\int_{S_{p}} \sigma (r \times n) dS = -\int_{S_{p}} r \times \nabla p dV, \]  

for the numerical integration.

The surface integral for the viscous term \( \sigma = \mu (\nabla u + (\nabla u)^{T}) \),

\[ F_{v} = \int_{S_{p}} \sigma \cdot n dS, \quad T_{v} = \int_{S_{p}} r \times (\sigma \cdot n) dS, \]  

is written for each component as

\[ F_{\sigma1} = \int_{S_{p}} \sigma_{1n_{1}} dS, \quad T_{\sigma1} = \int_{S_{p}} r_{1} \sigma_{1n_{1}} dS. \]  

As \( n_{1} dS \) is the projected area of \( dS \) in \( x_{1} \) direction, we calculate the numerical integration summing up for each span of mesh as presented two-dimensionally in Fig.2. Velocity components are located in the staggered arrangement as shown in Fig.2: arrow for velocity components, \( \circ \) for diagonal components of stress and \( \bigcirc \) for shear components for stress.

![fig2](image)

Fig. 2 Elements for surface integral of \( \sigma_{1n_{1}} \) component in viscous stress

2.5 Numerical scheme for particle-fluid interaction

This section summarizes a numerical procedure to advance in time from a flow fields \( u^{(n)} \), \( \rho^{(n)} \) in couple with particle motion \( v^{(n)} \), \( \omega^{(n)} \) given at \( t_{n} = n \Delta t \) to those at \( t_{n+1} \).

[1] By calculating each term in the Navier-Stokes equation of motion except for pressure gradient, the fractional step velocity is obtained by the second-order Adams-Bashforth scheme

\[ u^{*} = u^{(n)} + \frac{\Delta t}{2} [3H^{(n)} - H^{(n-1)}] \]  

where \( H \) contains convective term, viscous term and the external force term as

\[ H = -(u \cdot \nabla) u + \frac{1}{\rho} \nabla \cdot \sigma + g. \]  

[2] The fractional step velocity and angular velocity of each particle is also obtained without the contribution of pressure,

\[ v^{*} = v^{(n)} + \frac{\Delta t}{2m_{p}} [3(F_{v} + G)^{(n)} - (F_{v} + G)^{(n-1)}], \]  

\[ L^{*} = L^{(n)} + \frac{\Delta t}{2} [3(T_{v} + N)^{(n)} - (T_{v} + N)^{(n-1)}], \]  

\[ \omega^{*} = I_{p}^{(n)} + L^{*}. \]  

The fractional step velocity given by Eq.(16) at the grid point inside a particle is replaced by

\[ u^{*} = u^{*} + r \times \omega^{*}. \]  

[3] Using the fractional step velocity given by Eqs.(16) and (21), the Poisson equation for pressure is numerically solved as if the entire domain is occupied by fluid,

\[ \frac{1}{\rho} \nabla^{2} p^{(n+1)} = \frac{1}{\Delta t} \nabla \cdot u^{*}. \]  

[4] Adding the pressure contribution to the fractional step velocity completes an advancement to a new time step of fluid flow

\[ u^{(n+1)} = u^{*} - \frac{\Delta t}{\rho} \nabla p^{(n+1)}. \]  

The finite-difference system should be formulated to satisfy \( \nabla \cdot u^{(n+1)} = 0 \) everywhere in the computational domain at the stage of Eq.(23).

[5] Adding the contribution of pressure renews the particle motion,

\[ p^{(n+1)} = p^{*} + \Delta F_{p}^{(n+1)}, \]  

\[ L^{(n+1)} = L^{*} + \Delta T_{p}^{(n+1)}, \]  

\[ \omega^{(n+1)} = I_{p}^{(n)} + L^{(n+1)}. \]  

[6] Each particle is shifted by the Crank-Nicholson scheme,

\[ x^{(n+1)} = x^{(n)} + \frac{\Delta t}{2} [u^{(n)} + u^{(n+1)}]. \]  

Finally in the time advancement, the velocity at the grid inside the solid particle is adjusted by

\[ u^{(n+1)} = u^{(n+1)} + r \times \omega^{(n+1)}. \]
3. Computational Results

As the first step for the accuracy test of our method, we applied it to a two-dimensional flow past a circular cylinder and a three-dimensional flow past a sphere. Obstacles are fixed in a uniform stream. Thus we can skip the procedure of Eqs. (18)–(20) and (24)–(27) in the above mentioned scheme because they are concerning to the particle motion. Equations (21) and (28) which give the velocity inside a solid body are reduced to be $u^* = 0$ and $u'(n+3) = 0$ at all instances.

A result for the two-dimensional flow past a cylinder in a uniform stream was reported elsewhere[20]. The drag coefficient and the Strouhal number due to Karman vortices were in good agreement with experimental data in the Reynolds number up to around 500.

In this section, we focus our attention on the numerical result for a three-dimensional flow around a sphere fixed in the uniform stream. Figure 3 summarizes our computational condition. The numbers of grid points are $N_x = 120$ in the streamwise direction and $N_y = N_z = 48$ in the lateral directions, respectively. Each cell is a cube of the side length $h = 0.1$. The boundary conditions are as follows: a uniform stream at the inflow boundary, an approximation of $\partial u/\partial x = 0$ and a uniform pressure at the outflow boundary, and the periodic boundary condition for lateral directions. These boundary conditions are slightly different from those in a usual simulation for this flow field. As it is just a pre-computation for turbulent flows including many particles, we used a computational code for this purpose with the least modification. The ratio between a particle diameter $d_p$ and the grid width $h$ is an index of grid resolution. Here we tested three steps of resolutions as $d_p/h = 11, 8$ and 5. Although the grid resolution is getting poorer for smaller particle, the influence of finite domain is reduced in such a case since the ratio between the periodicity length and $d_p$ becomes larger.

Figure 4 compares computational results for the drag coefficient $C_D$ with experimental data collected by Schlichting, Fig. 1.5[21]. For the Reynolds number range in our simulation, $4 \leq Re \leq 600$ based on the relative velocity and the particle diameter, the drag coefficient by our method agrees reasonably with the experimental result. Referring to the experimental observation[22], the influence of grid resolution was tested for $Re = 4$ in steady range, $Re = 100$ in fluctuation range and $Re = 400$ in vortex shedding range, respectively. While the low Reynolds number case does not require a high resolution, it needs a large computational domain. This is a reason why the lowest resolution case $d_p/h = 5$ gave a reasonable agreement for $C_D$ at $Re = 4$. We confirmed that the high-resolution case gave closer agreement with experiment by expanding the computational domain by further computation. But the lowest resolution case $d_p/h = 5$ gave steady flow even for $Re = 100$ and $Re = 400$, aside from experimental observation. For resolutions of $d_p/h = 8$ and 11, numerical results are similar to each other and reasonably reproduced the wavy wake at $Re = 100$ and the vortex shedding at $Re = 400$. In addition to the condition shown in Fig. 3, we doubled the computational domain and the number of computational grid in $y, z$ directions. As a result, the qualitative agreement was improved by 4% for the case $Re = 400$ and $d_p/h = 8$.

Considering the above-mentioned results, we can conclude that our method with the resolution $d_p/h \geq 8$ has possibility to analyze the interaction between particle and fluid flow up to the Reynolds number of the order $10^5$. The result by Pan and Banerjee[14] shows a similar accuracy with ours, but the performance of their method is not clear especially in the Reynolds number range of vortex shedding.

---

Fig. 3 Computational domain and boundary condition for the simulation of flow around a sphere

Fig. 4 Drag coefficient of a sphere in a uniform stream
Figure 5 shows instantaneous flow fields around a sphere at the Reynolds number 500 by means of iso-surfaces of $\nabla^2 p$. There seems some noise attached to a sphere because we did not use a body-fitted grid system, while the noise is not observed in the wake region. An experimental observation\(19\) indicated that the hairpin-like vortices are shed from a sphere irregularly at the Reynolds number range $480 < Re < 650$. The pictures of $\nabla^2 p$ and the flow visualization by smoke are not completely identical. But our simulation seems to reproduce the shape of wake vortices and the unsteadiness in vortex shedding in comparison with the experiment\(19\). We also conducted the flow simulation using a body-fitted grid system for comparison. Wake vortices reproduced by the Cartesian grid and those by the body-fitted grid were almost same to each other. Although the Cartesian grid cannot represent smooth streamlines around a sphere, it is better in far field than the body-fitted grid since the grid resolution in the latter becomes rougher. Considering our major objective, that is the investigation of interaction between fluid turbulence and particle motion, the present scheme with a spatially-fixed Cartesian grid system is advantageous because it can reproduce the fluid force acting on the particle and the far wake vortices by a uniform resolution.

Next, we examine the accuracy for the evaluation of the budget equation of turbulence energy and Reynolds stress components, which is essential in the modeling of multiphase turbulence. The conservation equation of the turbulence kinetic energy $k = u_s u_h / 2$ is represented as

$$\frac{\partial k}{\partial t} + \frac{\partial u}{\partial x} \frac{\partial k}{\partial x} = -\frac{1}{\rho} \frac{\partial u}{\partial x} \frac{\partial k}{\partial x} + \nu \frac{\partial^2 k}{\partial x^2} - \frac{\partial u}{\partial x} \frac{\partial u}{\partial x}.$$

Taking the boundary conditions into account, an integral of the above equation from the inflow boundary as shown in Fig.3 to the streamwise location $X$ as well as in one periodic spans in $y, z$ directions derives

$$\frac{\partial K}{\partial x} = E_c + E_r + E_v + E_d.$$

Each term in this equation is

$$K = \int \nabla^2 p dV, E_c = \int \rho \left( \frac{\partial u}{\partial t} \frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} \frac{\partial u}{\partial y} + \frac{\partial u}{\partial z} \frac{\partial u}{\partial z} \right) dV,$$

$$E_r = \int \rho \left( \frac{\partial u}{\partial x} \frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} \frac{\partial u}{\partial y} + \frac{\partial u}{\partial z} \frac{\partial u}{\partial z} \right) dV,$$

$$E_v = \int \rho \left( \frac{\partial u}{\partial x} \frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} \frac{\partial u}{\partial y} + \frac{\partial u}{\partial z} \frac{\partial u}{\partial z} \right) dV,$$

where $S$ denotes the cross section perpendicular to the mainstream, $V(=SX)$ the integral volume. The terms in the right hand side of Eq.(30) are respectively $E_c$ the convection of energy, $E_r$ the work by pressure, $E_v$ the viscous diffusion and $E_d$ the dissipation rate. The convection term corresponds to the energy difference in between the inflow and outflow boundaries.

Figure 6 shows the time average of each term in Eq.(30) as a function of $X$ in the case $Re = 400$ and $dv/h = 11$. The viscous diffusion term is negligible in comparison with other terms because of the uniform inflow condition and no-gradient outflow condition. The total of time-averaged terms should be 0 in any
location $X$. The amount of error in time-averaged budget, indicated as 'Residual' in Fig. 6, is less than 6% of the dissipation rate or 4% of the pressure work at $X=10$. At $X=12$, it is less than 5% of the dissipation rate or 3% of the pressure work. The ratio of numerical residual in the budget to main terms is not so serious in our simulation. Thus we can conclude that our method has enough reliability in evaluating the contribution of wake vortices on the turbulence modulation due to particles.

Figure 7 shows the time evolution of each term in Eq. (30) integrated to the outflow boundary $X=12$. Figure 8 shows fluctuations in drag coefficient and lift coefficients (in the two lateral directions) at the corresponding time span as Fig. 7. The period of violent vortex shedding and the relatively steady stage appear alternatively as illustrated in Fig. 5. Such a characteristic could affect the instantaneous energy budget in the near-particle domain. The increase in drag coefficient detected in Fig. 8 is accompanied by the spiky increase in pressure work term $E_p$ in Fig. 7. The turbulence in the wake region increases but it is transferred out from the computational domain at the same time as shown by $E_c$ profile in Fig. 7. The viscous dissipation $E_d$, contrary to these terms, is almost constant. The turbulence energy induced by the shed vortices is not dissipated in the near-particle region but it is transferred to downstream. Thus the unsteady and non-equilibrium properties characterize the flow field caused by a particle with large relative Reynolds number. It could be parameterized by the time evolution of the drag force, since spiky increases in $dK/dt$, $E_p$ and $E_c$ are observed at the same time of sudden increase of $C_d$ in Fig. 8. These should be considered in turbulence modeling of particle-laden flows, especially in the unsteady calculation such as the large-eddy simulation (LES) with subgrid scale (SGS) models.

4. Conclusions

We proposed an efficient numerical scheme for the use of the direct numerical simulation of particle-laden turbulence. Applying our method to the flow around a sphere particle in a uniform stream, it satisfactorily reproduced the fluid force on the particle and the vortex shedding in the high Reynolds number region. Energy budget was also evaluated in an acceptable accuracy. We therefore think it could be a basis to investigate the influence of two-way interaction between fluid flow and particle motion. The grid resolution of $d_p/h=5 \sim 10$ may be reasonable for an actual application of our scheme to the multiphase flow including a large number of particles, considering both the efficiency and accuracy. In the case without vortex shedding, the resolution of $d_p/h=5$ could be enough. The resolution requirement to capture the wake vortices at the Reynolds number 500 or more.
based on the relative velocity is thought to be $d_S/h \geq 8$.

Concluding remarks about the characteristics of our method are as follows:

1. It is a kind of pure direct numerical simulation because it does not use any models for fluid force on the solid body. It is obtained by the surface integral of fluid stress.

2. The simplified CIP method allowed us to capture the flow around a particle with relatively rough resolution of grid at the surface.

3. Replacing the surface integral of pressure on the solid by the volumetric integral of the pressure gradient in the fluid flow eliminated the numerical error in momentum exchange between particle and fluid.

4. The residual in evaluating the energy budget by the resolution $d_S/h \approx 10$ is less than 6% of the dissipation rate of shed energy. It is enough for the analysis of turbulence modulation.

5. The application of our method is not restricted in the particle-laden flows. It is an efficient and accurate method by the fixed Cartesian grid for wide variety of usage; flow around a complex geometry, flow past many solid bodies, for example.

This study has been partially supported by the Grant-in-Aid for Scientific Research No. 09650189 from The Ministry of Education, Science and Culture.

References


