HYBRID VORTEX METHOD FOR HIGH REYNOLDS NUMBER FLOWS AROUND THREE-DIMENSIONAL COMPLEX BOUNDARY

Lingjia ZHAO, Hiroshi TSUKAMOTO *

Department of Biological Functions and Engineering, Graduate School of Life Science and System Engineering, Kyushu Institute of Technology, Kitakyushu 808-0196, Japan

---

Abstract

The hybrid vortex method, in which the vortex particle method was combined with the vortex sheet method, was extended to flows around a three-dimensional complex geometry boundary at high Reynolds number of $O(10^6)$. The computing domain was decomposed into an interior domain of vortex blobs and a thin numerical boundary layer of vortex sheets. The core radii of shedding blobs were related to the size of the vortex sheets. As the result of numerical experiments on the flow over a ship propeller, the hybrid vortex method was found to be acceptable for simulations of unsteady separated flows around a solid body at high Reynolds number, since the computed results showed reasonable agreement with the measured data.

Keywords: Hybrid vortex method; Numerical boundary layer; Vortex shedding; Fast multipole method; Propeller

---

* Corresponding author. Tel.: +81-93-695-6027, Fax: +81-93-695-6037
E-mail address: tsukamoto@life.kyutech.ac.jp (Tsukamoto)

1. INTRODUCTION

In the past three decades vortex methods have been discovered to be suitable for the computation of high-resolution simulation of unsteady and incompressible flows with a viscous boundary [1-4]. They are grid-free and thus eliminate the task of volumetric meshing of complex three-dimensional (3D) domains. This advantage is even more pronounced in situations where moving boundaries are encountered. Vortex methods are also self-adaptive, and hence are capable of concentrating computational elements dynamically in the regions where significant velocity gradients evolve.

Vortex methods have succeeded in simulations of two-dimensional (2D) bounded or unbounded flows. However, the conventional vortex methods display a few difficulties in simulations of 3D flows. The biggest difficulty lies in the enormous computational cost of evaluating the N computational particles interactions with a large number of particles in 3D simulations. Although this cost can be reduced by the fast multipole algorithms developed in the past two decades, it is impractical to simulate high Reynolds number (Re) flows with a large number of particles and without the help of advanced computers. In the study of the flow over a sphere at a low Re by Ploumhans et al. [3], the nascent vorticity at each boundary element was diffused from this element directly to the adjacent vortex blobs. The number of vortex blobs increases up to 616,000, 1,225,000, and 2,300,000 at $\mathrm{Re}=300$, 500, and 1000 respectively, since the size of vortex blobs should be less than $\sqrt{\nu\Delta t}$ to enforce convergence. Here, ν is kinematic viscosity, and Δt is time interval. The total run time would be approximately 230 hours on 64 processors of an HP V-Class system. The number of blobs increases rapidly with increasing Re and that makes the method impossible for simulations of high Re flow computations. The major purpose of this paper is to report on the efficient suppression of the increasing number of vortex blobs by increasing the size of the vortex blobs, and finally to reduce the computational cost in 3D engineering applications.

The idea of hybrid vortex methods derives from the random vortex sheet method by Chorin [5]. He did not employ the circular blobs in the conventional vortex methods, but adopted linear segment sheets to approximate the 2D boundary layer. The motion of sheets in the boundary layer was described by the Prandtl approximation of the Navier-Stokes equations. In the so-called hybrid vortex method the computational field is decomposed into an interior domain where the conventional vortex particle method is applied, and a thin user-specified numerical boundary layer where the random vortex method is used, as suggested by Chorin. It differs from the conventional vortex particle method where the total vorticity field is represented by a set of blobs carrying circulations. Nakanishi and Kamemoto [6, 7] applied the hybrid vortex method to the flow over a sphere at $\mathrm{Re} = 1000$. The 288 boundary elements used in their study, were much smaller than the 81,920 by Ploumhans et al. [3]. Thus the number of blobs shedding from the boundary could be decreased rapidly. Turkiyyah et al. [8] succeeded in predicting wind-induced pressures on 2D buildings using this method. Gharakhani et al. [9] extended this method to simulate the low Re flows in a simple 3D domain by employing the rectangular sheets in the numerical boundary layer. In these applications vortex blobs are shed not from the solid boundaries directly [3], but from the numerical boundary layer through the normal random walk. Once a sheet traverses the numerical boundary layer, it is converted into a vortex blob with a core equal to the diffusion displacement in the direction normal to the boundaries. Although these applications can give reasonable results by using blobs fewer than those in Ploumhans et al., the number of vortex blobs will still become more than a few million with increasing Re. Such a large number is unacceptable from the viewpoint of engineering applications. The reason for this large number is that the size of vortex blobs is of the order of the numerical boundary layer thickness, which becomes very small at high Re.

The goal of the present study is to improve the hybrid vortex method to simulate the high Re engineering-type flows. The presented method differs from the ones introduced in the references above on two points. First, the core radii of shedding blobs are related to the size of vortex sheets or the boundary elements, in terms of the relationship between the core radii of the blobs and the numerical boundary layer thickness. The shedding vortex blobs increase in size in the wake, and thus the number of blobs becomes smaller than those in the other vortex methods. Consequently, the hybrid vortex method can be applied to high Re flows in engineering applications. Second, the hybrid vortex method has succeeded in simulations of 3D flows around simple geometries where the shape of vortex sheets is a simple 2D plane [6, 7, 9]. In this study, the flow over a highly skewed ship propeller of a complex 3D curved surface was computed [10], where the body-fitted coordinate (BFC) system was used for the transformation from the physical space to the computational one. Circular sheets were used with respect to the BFC system to fit the complex boundary instead of rectangular sheets.

The following section presents the basic equations and numerical formulation for hybrid vortex method. Section 3 reviews the fast multipole method used to reduce the run-cost. Section 4 illustrates the performance of the method with several numerical examples.

2. NUMERICAL FORMULATION

In this section the governing equations for the hybrid vortex method and its numerical formulation for simulations of high Re flows around 3D complex boundary are presented.

For a 3D viscous and incompressible flow within a wall-bounded domain with boundary surfaces S, the governing equations may be written in the vorticity transport form of the Navier-Stokes equations

$$\frac{\partial \tilde{\omega}}{\partial t} + \left(\tilde{u} \cdot \nabla\right)\tilde{\omega} = \left(\tilde{\omega} \cdot \nabla\right)\tilde{u} + \nu \nabla^2 \tilde{\omega} \qquad (1)$$

$$\nabla \cdot \tilde{u} = 0 \tag{2}$$

$$\tilde{\omega} = \nabla \times \tilde{u} \tag{3}$$

$$\tilde{u}(\tilde{x}, t) = \tilde{u}_S \qquad \text{for} \quad \tilde{x} \in S \tag{4}$$

where $\tilde{x}$, $\tilde{u}$, $\tilde{\omega}$ and $\tilde{u}_S$ denote the position, velocity, vorticity and the velocity on the boundary surfaces S, respectively.

In the hybrid vortex method, the solution of the above N-S equations will be obtained by decomposing the computational field into an interior domain away from the solid walls and a thin numerical boundary layer adjacent to the solid walls (Fig.1). The vortex particle method is used in the interior domain for the solution of Equations (1) to (3) with a no-penetration boundary condition. On the other hand Prandtl approximation of the boundary layer is used to resolve the same equations with a no-slip boundary condition.

2.1 Interior Domain and Vortex Blobs

Flow in the interior domain will be computed using the grid-free vortex particle method whose details are given in references [1] and [3].

The flow field in the interior domain is bounded by the numerical boundary layer shown in Fig.1. The no-penetration condition is set at the edge of the layer so that no fluid can pass through the interface between the interior domain and numerical boundary layer. The normal component of velocity $(u_\infty, v_\infty, w_\infty)$ is zero at the edge of the layer, $w_\infty = 0$, so that the no-penetration condition may be satisfied (Fig.1). In order to generate a potential flow that cancels the normal component of the velocity induced by the vortex blobs and the incident flow, the boundary sources are distributed for all elements m on the body and satisfy a Neumann boundary condition.

$$\frac{1}{2}\sigma_m + \frac{1}{4\pi}\int_S \frac{\sigma_n}{x_{mn}^3}\tilde{x}_{mn} \bullet \tilde{n}_m dS_n + \tilde{W} \bullet \tilde{n}_m = 0 \tag{5}$$

where σ is the boundary source, $\tilde{n}$ is normal direction and $\tilde{W}$ is velocity induced by the vortex blobs and the incident flow.

In the interior domain, position $\tilde{x}_i$ and circulation $\tilde{\alpha}_i$ of blob i are updated at each time interval. The position $\tilde{x}_i$ is governed by the following equation

$$\frac{d\tilde{x}_i}{dt} = \tilde{u}_i \tag{6}$$

where the velocity $\tilde{u}_i$ of each blob consists of three parts, induced by the other blobs, the boundary source in Equation (5) and the incident flow. The circulation change due to 3D stretch can be expressed by

$$\left.\frac{d\tilde{\alpha}_i}{dt}\right|_{strech} = \left[\nabla \tilde{u}_i\right]^T \bullet \tilde{\alpha}_i \tag{7a}$$

which offers the advantage of conserving the total circulation. The circulation change due to viscous diffusion is based on the technique of particle strength exchange (PSE) [1, 3]. In the PSE algorithm the evolution equation for the blob circulation becomes

$$\left.\frac{d\tilde{\alpha}_i}{dt}\right|_{PSE} = \frac{\nu}{\delta_i^2}\sum_j \left(V_i\tilde{\alpha}_j - V_j\tilde{\alpha}_i\right)\psi_{\delta_i}\left(\tilde{x}_i - \tilde{x}_j\right) \tag{7b}$$

where δ_i and V_i are the core radius and the volume of the i th blob respectively, and ψ_{δ_i} is the cutoff function.

In order to improve the accuracy of time integration, the second-order Adams-Bashforth scheme is used for equations (6) and (7).

$$\tilde{x}_i^{k+1} = \tilde{x}_i^{k} + \Delta t\left(\frac{3}{2}\tilde{u}_i^{k} - \frac{1}{2}\tilde{u}_i^{k-1}\right) \tag{8}$$

$$\tilde{\alpha}_i^{k+1} = \tilde{\alpha}_i^{k} + \Delta t\left.\left(\frac{3}{2}\frac{d\tilde{\alpha}_i^{k}}{dt} - \frac{1}{2}\frac{d\tilde{\alpha}_i^{k-1}}{dt}\right)\right|_{stretch+PSE} \tag{9}$$

The attractive advantage of vortex methods is that the pressure can be dependently computed from the vorticity and velocity field, when needed, by solving a Poisson equation with Neumann boundary conditions [11, 12]. After the vorticity and velocity field are determined in the above procedures, the unsteady pressure field may be obtained from the following equation (10), where B is the Bernoulli function, the coefficient β is 1 for $\tilde{x}$ inside the flow field, 0.5 for $\tilde{x}$ on a smooth boundary and 0 for $\tilde{x}$ outside the flow field. If the flow Re is sufficiently high, the convection term is considered much larger than the viscous diffusion term, and thus the second term in the right hand side of equation (10) may be neglected in the computation.

$$\begin{aligned}
&\beta B(\tilde{x})+\frac{1}{4\pi}\int_S B(\tilde{x}_0)\frac{\tilde{x}-\tilde{x}_0}{\left|\tilde{x}-\tilde{x}_0\right|^3}\bullet\tilde{n}\left(\tilde{x}_0\right)dS_0\\
&\qquad=\frac{1}{4\pi}\int_S\frac{\partial\tilde{u}(\tilde{x}_0)}{\partial t}\bullet\tilde{n}\left(\tilde{x}_0\right)\frac{1}{\left|\tilde{x}-\tilde{x}_0\right|}dS_0\\
&\qquad+\frac{1}{4\pi}\nu\int_S\left\{\tilde{\omega}(\tilde{x}_0)\times\tilde{n}\left(\tilde{x}_0\right)\right\}\bullet\frac{\tilde{x}-\tilde{x}_0}{\left|\tilde{x}-\tilde{x}_0\right|^3}dS_0\\
&\qquad-\frac{1}{4\pi}\int_V\left\{\tilde{u}(\tilde{x}_0)\times\tilde{\omega}\left(\tilde{x}_0\right)\right\}\bullet\frac{\tilde{x}-\tilde{x}_0}{\left|\tilde{x}-\tilde{x}_0\right|^3}dV_0
\end{aligned}\tag{10}$$

2.2 Boundary Domain and Vortex Sheets

In the boundary domain, vortex blobs do not provide good approximations of the vorticity because the flow has a big velocity gradient in the normal direction that is not well captured by the symmetric blob representation. A thin user-specified "numerical boundary layer" is assumed, where Prandtl approximations are utilized. Approximation of vorticity in the numerical boundary layer consists of the segments parallel to the boundary. These segments are called vortex sheets [5].

2.2.1 No-slip boundary condition and sheets generation

To impose the no-slip condition along the solid boundaries, the boundaries are divided into 2D segments of width h.

Initially, there is no vorticity in the field and a potential flow is imposed on the geometry such that the normal flux through the solid boundaries may be zero. This in turn induces a slip velocity (u_∞, v_∞) on the edge of the boundary layer or on the body surface (the thin boundary layer thickness, b, can be omitted when computing the slip velocity). It differs from the prescribed boundary condition (u_b, v_b) (Fig.1). Such a velocity jump across the wall is expressed by "surface vorticity sheet" $\tilde{\gamma}(\tilde{x}, t)$ centered at the boundary element with an area equivalent to the boundary element and strength equal to the slip velocity. The boundary element is called the "mother boundary element" for a nascent sheet.

$$\tilde{\gamma}(\tilde{x}, t) = (\gamma_x, \gamma_y, \gamma_z) = (-(v_\infty - v_b), (u_\infty - u_b), 0) \tag{11}$$

In a simple 3D domain, the solid wall can be carpeted by a set of rectangular elements [9]. However, the boundary surface in a complex 3D domain has to be discretised into a set of trapeziums. If a nascent sheet inherits the shape of its mother boundary element, it is difficult to calculate the overlap coefficient between two sheets. In this study 2D circular sheets with the same area as their mother boundary elements were used instead of the rectangular sheets. Consequently, all sheets are similar, though they hold different areas.

2.2.2 Sheet evolution

Once the vortex sheet is generated on the solid wall, its evolution is approximated within the numerical boundary layer by the Prandtl equations [13, 14].

$$\frac{\partial \tilde{\omega}}{\partial t} + \left(\tilde{u} \cdot \nabla\right)\tilde{\omega} = \frac{1}{\mathrm{Re}} \frac{\partial^2 \tilde{\omega}}{\partial z^2} \tag{12a}$$

$$\nabla \cdot \tilde{u} = 0 \tag{12b}$$

$$\tilde{\omega} = (\omega_x, \omega_y, \omega_z) \cong (-\frac{\partial v}{\partial z}, \frac{\partial u}{\partial z}, 0) \tag{12c}$$

$$\tilde{u}(x, y, z = 0, t) = (0, 0, 0) \tag{12d}$$

$$\tilde{u}(x, y, z = b, t) = (u_\infty, v_\infty, 0) \tag{12e}$$

Here all variables are defined with respect to the local coordinates system $\left(x, y, z\right)$, in which the xy plane is tangential to the body surface, z is assigned to be normal to the body surface and to point into the flow interior, and $z = 0$ represents the wall surface (Fig.1). Vorticity stretch is assumed to be negligible within the boundary layer.

For a given sheet distribution within the layer, the velocity components $\tilde{u}(u, v, w)$ in the local coordinates system can be approximated by the following equations evolved from equation (12) [5, 9]

$$u(\tilde{x}_i,t)=u_\infty(x_i,y_i,b,t)-\frac{1}{2}\tilde{\gamma}_y(\tilde{x}_i,t)-\sum_{\substack{j=1\\ j\neq i}}^{N_T}\tilde{\gamma}_y(\tilde{x}_j,t)\varphi_j(x_i,y_i)H(z_j-z_i) \qquad (13a)$$

$$v(\tilde{x}_i,t)=v_\infty(x_i,y_i,b,t)-\frac{1}{2}\tilde{\gamma}_x(\tilde{x}_i,t)-\sum_{\substack{j=1\\ j\neq i}}^{N_T}\tilde{\gamma}_x(\tilde{x}_j,t)\varphi_j(x_i,y_i)H(z_j-z_i) \qquad (13b)$$

$$w(\tilde{x}_i,t)=0 \qquad (13c)$$

for $i=1,2,...,N_T$

where H is the Heaviside step function, N_T is the total number of vortex sheets and $\varphi_j(x_i,y_i)$ is the overlap coefficient which denotes the area A_i of sheet i covered by sheet j

$$\varphi_j(x_i,y_i)=\frac{A_i\cap A_j}{A_i}$$

In equation (13) the assumption of $w(\tilde{x}_i,t)=0$ is valid for high Re flows. The normal component w is much smaller than the tangential components (u,v) and the normal viscous diffusion in a sufficiently high Re flow. In this case the convection of sheets in the boundary layer can be regarded as 2D motion parallel to the boundary surface, and the normal displacement is induced only by viscous diffusion. With decrease in Re, however, the normal component becomes large and cannot be neglected.

In order to describe the convection motion of vortex sheets around complicated geometry, the body-fitted coordinate (BFC) system is adopted to transform the 3D body surface into a 2D plane (ξ,η) in the computational space. The transformation from the local coordinate velocity components (u,v,w) to the contravariant velocity (U,V) is given by

$$U = \xi_x u + \xi_y v + \xi_z w \tag{14a}$$

$$V = \eta_x u + \eta_y v + \eta_z w \tag{14b}$$

where, subscripts denote differential.

The grid-free tile solution within the numerical boundary layer for the sheet i located at (ξ_i,η_i) is governed by the following equation.

$$\xi_i^{k+1} = \xi_i^k + U_i \Delta t \tag{15a}$$

$$\eta_i^{k+1} = \eta_i^k + V_i \Delta t \tag{15b}$$

Then the position $\tilde{x}_i$ in the local coordinate system (x,y,z) is transformed from the BFC system (ξ,η).

$$x_i^{k+1} = x(\xi_i^{k+1},\eta_i^{k+1}) \tag{16a}$$

$$y_i^{k+1} = y(\xi_i^{k+1},\eta_i^{k+1}) \tag{16b}$$

$$z_i^{*} = z(\xi_i^{k+1},\eta_i^{k+1}) \tag{16c}$$

$$z_i^{k+1} = \left| z_i^* + \varsigma_i(\Delta t) \right| \tag{16d}$$

where a random variable ς is drawn from a Gaussian distribution with mean 0 and variance $2\nu\Delta t$. The asterisk pertains only to the z direction, implying that viscous diffusion is in the direction normal to the wall.

2.3 Vorticity shedding and numerical procedure

2.3.1 Vorticity shedding from the numerical boundary layer

Vortex sheets carry concentrations of vorticity and the velocity field is determined by their locations and concentrations. At each time step, they are diffused in the direction normal to the boundary as equation (16d) and convected in the direction tangential to the boundary as equations (15) and (16a-c). Once a sheet (sheet 1 in Figure 2) traverses the numerical boundary layer in the direction normal to the boundary, it is converted into a spherical vortex blob with a volumetric vorticity vector equal to that of the sheet. Furthermore, any sheets (sheet 2 in Figure 2) that jump below the wall are reflected back into the field [15].

The mode of vortex shedding from the boundary to the wake is assumed to be dependent only on viscous diffusion. A vortex sheet (sheet 3 in Figure 2) will not be converted into a blob when it is convected and shed from a discontinuous corner of the body surface. It will then be replaced by a new sheet created at the next time step.

Vortex blobs shedding from the numerical boundary layer are convected by the vortex particle method described in Section 2.1. A blob will be ignored when it enters into the numerical boundary layer from the interior domain. As a result of the motion of blobs and sheets, a net slip velocity is induced on the solid boundaries, which is canceled by generating new sheets there.

2.3.2 The determination of the size of shedding blobs

In vortex methods the overlapping of vortex blobs shown in Figure 1 is a critical factor for the accuracy in the computation of both diffusion and velocity [1], The overlapping requirement imposes that the size of blobs surrounding each blob should exceed the inter-blob spacing. Then the number of blobs shedding from the boundary can be decreased with increasing the size of blobs.

There is no criterion for determining the size of blobs shedding from the boundary layer. As we have seen above, Ploumhans's method for determination of blob core is impractical for simulations of high Re flows without the help of advanced computers. Kamemoto [7] enlarged the core radius of blobs to be of the order of the boundary layer thickness and somewhat suppressed the increasing number of blobs with increasing Re. But it is still impractical for simulations of high Re flows because the boundary layer thickness is very small, and hence the number of blobs is huge.

The core radius of blobs in the wake could be enlarged further because the wake has a smaller vorticity gradient than the domain adjacent to the boundary surface where sheets can provide better approximation than blobs. In other words, there would be no necessity to use rather small size of blobs outside the numerical boundary layer.

In this paper the core radius in the wake is related to the size of sheet, since circulation carried by a spherical vortex blob is equal to the one carried by a sheet, $\tilde{\alpha} = \tilde{\gamma} A(h)$ (where h and A are the size and area of a sheet respectively) and also equal to the volumetric integral of vorticity vector itself, $\tilde{\alpha} = \tilde{\omega}_0 V(\delta)$ (where $\tilde{\omega}_0$ and V are the averaged vorticity vector and volume of a blob). Considering the computational cost and accuracy, we take the core radius as half the length of sheet. A convergent result can be obtained as long as the size of sheet (or the size of boundary elements) is sufficiently small.

However, this determination of blob core would be improper for low Re flow because of the lower computational accuracy. The size of blobs should be of the order of $\sqrt{\nu \Delta t}$ suggested by Ploumhans, or of the order of the boundary layer thickness suggested by Kamemoto for low Re flows where the number of blobs is not huge.

2.3.3 Numerical procedure

The simulation procedure consists of the following steps:

1) Compute boundary sources satisfying the no-penetration condition on the boundary.

2) Compute the velocity field in the interior domain due to blobs and sources.

3) Compute the tangential velocity at the edge of boundary layer due to blobs and sources.

4) Compute the velocity of each sheet in the boundary layer.

5) Compute the net slip velocities on the solid walls and create new sheets to satisfy the no-slip condition.

6) Update the position of blobs and sheets.

7) Examine the positions of blobs and sheets, omit blobs entering into the boundary layer and execute the transition from sheet to blob.

8) Compute the pressure field.

9) Iterate from step 1 until the calculation has converged.

The computational parameters, including the numerical boundary layer thickness b, the maximum sheet strength MSS, the size of boundary elements (or vortex sheets) h and the radii of blobs δ are important factors for the convergence of computation: Fishelov [14] suggested that the numerical boundary layer thickness should be several times the standard deviation of the random walks of the vortex sheets; $b = (1.0 \sim 3.0) \times \sqrt{2\Delta t/\mathrm{Re}}$, so that the probability of any vortex sheets moving outside the boundary layer is very small in one walk. Therefore newly created vortex sheets will stay near the mother boundary elements for a number of time steps before they move into the interior flow and become vortex blobs. The MSS can be determined so that the degree of refinement of the velocity in the normal direction must be of the same order as the degree of refinement in the tangential direction. MSS also determines the maximum tangential velocity of sheets in the boundary layer, $u_{\max}$. The size of boundary element should be chosen so that the geometry is faithfully reproduced and sheets move downstream at no more than one element per time step, $\Delta t \cdot u_{\max} \leq h$. The radii of blobs satisfy $\delta = h/2$.

3. FAST MULTIPOLE METHOD (FMM)

A fundamental issue of *N*-body problems is the ability to evaluate all N^2 pairwise interactions in large ensembles of particles, i.e., expressions of the form

$$\tilde{u}_i(\tilde{x}_i) = \frac{1}{4\pi} \sum_j \frac{\tilde{\alpha}_j(\tilde{x}_j) \times (\tilde{x}_i - \tilde{x}_j)}{\left|\tilde{x}_i - \tilde{x}_j\right|^3}$$

for the velocity field $\tilde{u}$ induced by the vorticity field $\tilde{\alpha}$ in vortex methods.

In the past two decades the fast multipole method (FMM) has been developed as a technique to reduce the computational cost and memory requirements in solving *N*-body problems. The FMM was initially introduced by Rokhlin [16] as a fast solution method for integral equations, and then was applied to *N*-body problems by Greengard [17]. A new version of FMM introduced by Greengard [18] accelerates this kind evaluation dramatically from $O(N^2)$ to $O(N \log N)$. Cheng [19] developed Greengard's FMM to an adaptive scheme based on an adaptive data structures.

In this study we have reduced the run-time and have run up to 200,000 vortex blobs on a Dell8250 workstation using Cheng's FMM.

4. NUMERICAL RESULTS

4.1 Flow over a cube

The validity of the present method was checked for the flow over a stationary cube, whose drag coefficient at fully turbulent flows ($10^4 \le \mathrm{Re} \le 10^6$) is 1.05 [20]. Here the drag coefficient C_D is defined as:

$$C_D = \frac{F_d}{\frac{1}{2}\rho V_p^2 a^2}$$

where ρ, V_p, a, and F_d denote the density, the uniform velocity, the cube side, and the drag force on the cube, respectively. This flow was chosen because; (1) the convection evolution of sheets is 2D plane motion in the numerical boundary layer, (2) it is not necessary to impose the BFC system, (3) the boundary surface can be discreted into uniform elements, (4) the overlap between two sheets can be calculated exactly.

Calculation was done for $\mathrm{Re} = 10^5$. The non-dimensional time step defined as $\Delta T = {\Delta t \cdot V_p}/{a}$, was taken as $\Delta T = 0.05$. The numerical boundary layer thickness was set to $0.01a$ ($\approx 3 \cdot \sqrt{2\Delta t / \mathrm{Re}}$). Each side of the cube was uniformly discretized into 100 square elements so that vortex sheets may move downstream at no more than one element per time step. The total number of vortex blobs was up to $40,000$. Using Cheng's FMM the run-time was about 50 hours on a Dell8250 computer with a CPU clock rate of 3.06 GHz and RAM of 1 GB.

Figure 3 shows the time histories of the calculated drag coefficient. The drag force is calculated by integrating the pressure on the surface of the cube. The calculated time-averaged drag coefficient is about 0.92, which shows reasonable agreement with the measured one. This indicates that the hybrid vortex method is valid for simulations of unsteady separated wake behind a solid body in high Re flow.

Figure 4 shows the effect of the core radius of the blobs on the number of blobs and the computational cost for uniform flow over a cube. Figure 5 shows the effect of the core radius of the blobs on the solution of the drag coefficient. Here, computations were done under conditions of $\mathrm{Re} = 10^5$, $\Delta T = 0.05$ and the same boundary discretizations. The number of blobs as well as the computation time increases remarkably with decreasing blob size, as these figures show. Once the size of the blobs is chosen to be small enough, the solution of the drag coefficient becomes independent on the size of the blobs. Although the smaller core radius gives a relatively accurate result, it leads to an increase in the number of vortex blobs and the computational cost. Obviously, it is not practical to apply huge number of blobs in engineering applications because of the expensive computational cost. Based on the balance of the computational cost and accuracy, the core radius was taken as half the size of sheet (or the size of the boundary element) in the present study.

4.2 Flow over a sphere

The validity of the present method was also checked for the flow over a stationary sphere. The goal was to assess the accuracy of the modeling approach in predicting quantities such as the pressure distributions around the sphere and the drag forces. Additionally, the present method can capture the separation line automatically. The separation line cannot be predicted in advance, whereas the virtual separation lines can be set at the corners for the flow over a cube.

The calculation was computed for $\mathrm{Re} = 10^5$ and 4.35×10^5, above and below the critical Re. The non-dimensional time step was defined as $\Delta T = {\Delta t \cdot V_p}/{a}$, where a denotes the diameter of sphere, and it was taken as $\Delta T = 0.01$. The numerical boundary layer thickness was set to $0.01a$. The surface of sphere was discretized into 1,206 elements. The total numbers of vortex blobs were up to $78,000$ and 96,000. Using Cheng's FMM the run-times were about 110 and 150 hours on a Dell8250 computer with a CPU clock rate of 3.06 GHz and RAM of 1 GB.

The flow over a sphere is a benchmark for bluff-body flows with many numerical and experimental data. The most well known feature of the flow over a sphere is the occurrence of the drag crisis around a critical Re of 3.7×10^5 [21]. At Re higher than the critical one, the drag coefficient drops from values near 0.4-0.5 prior to the drag crisis to around 0.07-0.09 within a small range of Re. Figure 6 compares the calculated drag coefficients by the present method with the measured ones from Schlichting [22]. The present method predicts this sudden drop in the drag force. The calculated time-averaged drag coefficient, 0.44, agrees with the measured one, 0.42, for the subcritical state of $\mathrm{Re} = 10^5$ in which the physical boundary layer is laminar. On the other hand, the agreement between the calculated drag coefficient, 0.12, and the measured one, 0.09, is somewhat less satisfactory for the supercritical state of $\mathrm{Re} = 4.35 \times 10^5$ in which the physical boundary layer becomes turbulent. Figure 7 shows the surface pressure distributions for the subcritical and the supercritical states. The calculated pressure coefficient at the upstream stagnation point does not go to 1.0 because the pressure distribution over a boundary element is assumed to be uniform in this study, and the pressure is taken to be the average over the boundary element centered at the stagnation point. Calculated Cp shows the difference from the measured one at the downstream side for the supercritical case. This is because the energizing action of the outer flow is much greater on the turbulent boundary layer than in the laminar case [22]. Because of the turbulent mixing motion, more vortex blobs are taken into the numerical boundary layer near the downstream side of the sphere. It may be improper to eliminate these vortex blobs rudely, since they contribute to the total vorticity field. More attention should be paid to the vortex blobs near the downstream side when the physical boundary layer becomes turbulent.

The fact that calculated results for the flow over a sphere as well as a cube show reasonable agreement with measured ones indicates that the present method is suitable for unsteady separated flows around a solid body at high Re. However, further improvement may be necessary for the separation of the turbulent boundary layer.

4.3 Flow over a ship propeller

The present hybrid vortex method was applied to a highly skewed ship propeller [10] in a uniform flow (Fig.8). This flow field is charactered by high Re and a complex geometry boundary. The determination of the core radius of vortex blobs by this study is expected for the simulation of this engineering-type flow at high Re. The employment of BFC can accurately track the evolution of vortex sheets in the complex boundary layer. Moreover, the unsteady wake and the propeller performance are often associated with the vortices shed from the leading edge and the trailing edge of the blade. The present method should be applicable as long as the problems related with the unsteady flow separation are concerned.

The specifications of the model propeller and measured data are provided by Nagasaki R & D Center, Mitsubishi Heavy Industries LTD, as shown in Tables 1 and 2. Here,

Forward coefficient $$J = \frac{V_p}{n \cdot D_2}$$

Thrust coefficient $$K_T = \frac{T_h}{\rho \cdot n^2 \cdot D_2^4}$$

Torque coefficient $$K_Q = \frac{\mathrm{Q}}{\rho \cdot n^2 \cdot D_2^5}$$

Efficiency $$e_p = \frac{J \cdot K_T}{2\pi K_Q}$$

Where ρ denotes density, n, D_2, V_p, T_h and Q denote the rotational speed, the diameter, the forward speed, the thrust, and the torque of the propeller, respectively. The T_h and Q can be calculated by integrating the pressure on the propeller surface.

The Re based on the propeller diameter, D_2, and the circumferential velocity, $\pi n D_2$, was set to be 2.36×10^6. The non-dimensional time step defined as $\Delta T = 360 \cdot \Delta t \cdot n$, which denotes propeller rotation angle per step, was taken as $\Delta T = 1.08$. The surface of each blade was modeled with about 600 elements so that the body surface can be well reproduced and vortex sheets may move downstream no more than one element per time step. The numerical boundary layer thickness at $\mathrm{Re} = 2.36 \times 10^6$ and $\Delta T = 1.08$ is about 1% of the square root of the boundary element area according to Fishelov [14]. However, it was set to be a tenth of the square root of the boundary element area, because the suggested thickness is too thin.

Using Cheng’s FMM, computation was made up to 2×10^5 vortex blobs. The run-time was about 230 hours on a Dell8250 computer with a CPU clock rate of 3.06 GHz and RAM of 1 GB. Figure 9 shows the time histories of the total number of vortex blobs at $J = 0.60$, 0.70 and 0.80. In all computed cases the total number of blobs reaches a plateau after the propeller runs about 5 revolutions. Figure 10 shows the instantaneous distribution of vortex blobs shed from a blade when the computation converged ($J = 0.75$, $T = 12$). It can be seen that vortex blobs shed mostly from the propeller tip and mid of the suction side.

Figure 11 shows the time variation in the thrust coefficient and the torque coefficient at $J = 0.75$. The other cases also become steady after the propeller runs about 4 revolutions.

Table 2 and Figures 12 to 14 show the comparison of the calculated time-averaged thrust coefficient, torque coefficient and efficiency with the measured ones. There exists the best efficiency point where the calculated K_T and K_Q show the minimal differences from the measured ones. Calculated and measured K_T as well as K_Q show bigger differences at smaller and larger forward coefficient. Those are thought to be due to more violent turbulent behavior which is difficult to describe in the present calculation.

Figure 15 shows the time-averaged pressure distributions on the suction (SS) and pressure surface (PS) of the propeller for $J = 0.60$, 0.70 and 0.80 . Pressure coefficients in this figure are normalized by the circumferential velocity of the propeller:

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho\left(\pi n D_2\right)^2}$$

There is a visible low pressure region near the tip along the leading edge of the propeller, where attention should be paid for the problem of cavitation.

5. CONCLUSIONS

The hybrid vortex method has been extended to the 3D bounded flows with complex geometry boundaries at high Reynolds number. The numerical boundary layer on the body surface is expressed by a series of circular sheets. The core radius of blobs shedding from the boundary layer is determined by our original method as half the length of sheet or boundary element. Numerical experiments were conducted on the flows over a cube, a sphere and a ship propeller. Calculated results were compared with the corresponding measured ones, and the following conclusions were derived:

(1) The numerical boundary layer is valid for large velocity gradients in the boundary regions.

(2) Using the blob core radii related to the boundary elements, the number of vortex blobs can be reduced in number when compared with the conventional vortex particle method.

(3) As its application to the flow over a cube as well as a sphere showed, the hybrid vortex method is a computational method applicable to unsteady separated wake behind a solid body in high Re flow. However, it should be improved to simulate the separation of the turbulent boundary layer. With a decrease in Reynolds number, the determination of the blob core dimension can be problematic. Additionally, the assumption of the

normal component of sheet velocity, and the omission of the viscous term in equation (10), are no longer valid.

(4) By imposing the BFC system and circular sheet, the hybrid vortex method is effective to simulate the flow around complex geometry as the numerical results of the flow over a highly skewed ship propeller show.

ACKNOWLEDGMENT

The authors gratefully acknowledge the support of the Nagasaki Research and Development Center, Mitsubishi Heavy Industries, LTD for providing the geometry and the measured data of the model propeller.

REFERENCES

[1] Cottet GH, Koumousakos P. Vortex methods: theory and applications. Cambridge: Cambridge Univ. Press; 2000.

[2] Lewis RI. Vortex element for fluid dynamic analysis of engineering systems. Cambridge: Cambridge Univ. Press; 1991.

[3] Ploumhans P, Winckelmans GS, et al. Vortex methods for direct numerical simulation of three-dimensional bluff body flows. J. Comput. Phys. 2002; 178: 427-463.

[4] Wu JC, Thompson JF. Numerical solutions of time-dependent incompressible Navier-Stoked equations using an intergro-differential formulation. Computers and Fluids 1973; 1: 197-215.

[5] Chorin AJ. Vortex sheet approximation of boundary layers. J. Comput. Phys. 1978; 27: 428-442.

[6] Nakanishi Y., Kamemoto K. Numerical simulation of flow around a sphere with vortex blobs. J. Wind Engng. and Ind. Aero. 1992; 46: 363-369.

[7] Kamemoto K. On attractive features of the vortex methods. Computational Fluid Dynamics Review. 1995; 334-353.

[8] Turkiyyah G, Reed D, Yang J. Fast vortex methods for prediciting wind-induce pressures on buildings. Journal of Wind Engineering and Industrial Aerodynamics 1995; 58: 51-79.

[9] Gharakhani A, Ghoniem AF. Three-dimensional vortex simulation of time dependent incompressible internal viscous flows. J. Comput. Phys. 1997; 134: 75-95.

[10] http://www.nmri.go.jp/trans/Staff/kudo/database/SEIUN-pressure/

[11] Wang H. Experimental and numerical study of unsteady flow in a diffuser pump. Doctor Thesis, Kyushu Institute of Technology. 2001.

[12] Wang H, Tshukamoto H. Experimental and numerical study of unsteady flow in a diffuser pump at off-design conditions. ASME Journal of Fluids Engineering 2003; 125:767-778.

[13] Chorin AJ. Vortex models and boundary layer instability. SIAM J. Sci. Stat. Comput. 1980; 1(1): 1-21.

[14] Fishelov D. Vortex methods for slightly viscous three-dimensional flow. SIAM J. Sci. Stat. Comput. 1990; 11(3): 399-424.

[15] Puckett EG. A study of the vortex sheet method and its rate of convergence. SIAM J. Sci. Stat. Comput. 1989; 10(2): 298-327.

[16] Rokhlin V. Rapid solution of integral equations of classical potential theory. J. Comput. Phys. 1985; 60: 187-207.

[17] Greengard L. The rapid evaluation of potential fields in particle systems. Cambridge, MA: MIT Press, 1988.

[18] Greengard L, Rokhlin V. A new version of the fast multipole method for the Laplace equation in three dimensions. Acta Numerica. 1997; 6: 229-269

[19] Cheng H, Greengard L and Rokhlin V. A fast adaptive multipole algorithm in three dimensions. J. Comput. Phys. 1999; 155: 468-498.

[20] Holloway KA, Patharkar M. Fundamentals of gas management as applied to nonwovens production. International Nonwovens Journal 2004; 13(1): 27-32.

[21] Constantinescu G and Squires K. Numerical investigations of flow over a sphere in the subcritical and supercritical regimes. Physics of Fluids 2004; 16(5): 1449-1466.

[22] Schlichting H and Gersten K. Boundary layer theory. 8th edt. New York McGraw-Hill; 1999.

[23] Guermond JL and Lu HZ. A domain decomposition method for simulation

advection dominated, external incompressible viscous flows. Computers & Fluids 2000; 29: 525-546.

[24] Moriwaki Y, et al. Numerical analysis of flow around a ship propeller. JSME Pre-print. 2005; No.058-2.