Finite Difference Scheme for Solving Incompressible Viscous Flow around an Oscillating Body*

Motomu NAKASHIMA** and Kyosuke ONO***

This paper presents a finite difference scheme for solving incompressible viscous flow around an arbitrary oscillating body. This scheme is an extension of the SMAC method to a moving curvilinear coordinate grid. The scheme comprises two stages. At stage one, the new time step velocity is calculated in the old time step coordinates. At stage two, this calculated velocity is reobserved in terms of the new time step coordinates (rezoning). This scheme is applied to calculation of two-dimensional flow around an oscillating flat plate, which is a fundamental model of the fin movement of certain kinds of fish or cetaceans or the flutter of an airplane wing. The solution is compared with that from the discrete vortex method. In particular, the characteristics of thrust and propulsive efficiency of the oscillating flat plate as a propulsion mechanism show good agreement with those from the discrete vortex method.

Key Words: Computational Fluid Dynamics, Finite Difference Method, Incompressible Viscous Fluid Flow, SMAC Method, Rezoning, Discrete Vortex Method

1. Introduction

Many kinds of incompressible viscous flows (e.g., flows around fins of fishes or cetaceans⁽¹⁾, airplanes, fluid machinery, etc.) have been studied by mathematical approaches. Recently, however, flows around automobiles and structures, for example, which are difficult to solve by mathematical methods and are accompanied by oscillation, have become important problems to be solved. In general, flow around an oscillating body is quite different from that around a stationary body, even though the amplitude of oscillation is one or two percent of the size of the body⁽²⁾. Recently, the discrete vortex method has attracted attention as a powerful analytical method for flow around a moving body. In this method, two-dimensional incompressible viscous flow is regarded as a linear combination of potential flow and a series of point vortices. Sakata et al.⁽³⁾ developed a method in

which the potential flow around a body is represented by point vortices. With this method, it becomes easier to take account of the motion of the body. The flows around, for example, a savonius rotor⁽⁴⁾, and a Weis-Fogh mechanism⁽⁵⁾, have been analyzed with this method.

In the discrete vortex method, the strengths of shed vortices can be determined by Kelvin's theorem in the case where the separated field is confined within a small area, such as flow around a wing. However, in the case, for example, of flow around a circular cylinder, it is difficult to accurately determine the positions of the separation and the strengths of the shed vortices. In addition, it is necessary to introduce some model of fluid viscosity in order to solve viscous flow. In light of these problems, it is considered that the discrete vortex method is not sufficient to solve general flows accurately, although it allows solution of certain local problems or qualitative simulation.

On the other hand, various incompressible viscous flows have been analyzed by many researchers by means of the finite difference method. In this method the Navier-Stokes equation is solved directly, so that a complicated separated flow may be accurately

* Received 30th September, 1992. Paper No. 91-0972 A

** Graduate School, Tokyo Institute of Technology, 2-12-1 Oh-okayama, Meguro, Tokyo 152, Japan

*** Faculty of Engineering, Tokyo Institute of Technology

simulated. The MAC method⁽⁶⁾ and the SMAC method⁽⁷⁾, in which the velocity and the pressure of the fluid are used as variables to be solved, are typical methods. These methods are extended to solution of high-Reynolds-number flow using a body-fitted curvilinear coordinate grid so that accurate results can be obtained⁽⁸⁾. In the case of flow around an oscillating body, however, it is necessary to move the coordinate grid together with the body. Therefore, it is very important to develop a finite difference method which allows solution of the incompressible viscous flow around an oscillating body using a moving curvilinear coordinate grid, and to verify the accuracy of the method. In this paper, we propose a scheme which is an extension of the SMAC method to a moving curvilinear grid, and which can be applied to flow around an arbitrary-shaped oscillating body. Moreover, we apply this scheme to analysis of a two-dimensional flow around an oscillating flat plate⁽⁹⁾, which is a fundamental model of the fin movement of certain kinds of fish or cetaceans or the flutter of an airplane wing, and verify the accuracy of this scheme by comparing the results of its propulsive characteristics with those from the discrete vortex method.

Nomenclature

- x, y : Cartesian coordinates
 ξ, η : curvilinear coordinates
 Re : Reynolds number
 ρ : fluid density
 u, v : velocities of fluid
 U, V : contravariant velocities of fluid
 U^*, V^* : new contravariant velocities measured in terms of old coordinates
 $U_{\text{mesh}}, V_{\text{mesh}}$: contravariant velocities of coordinate grid
 p : pressure of fluid
 t : time
 Δt : time step
 n : number of time steps in finite difference method
 ∇ : nabla operator
 J : Jacobean $= x_\xi y_\eta - x_\eta y_\xi$
 $g_{11} = x_\xi^2 + y_\xi^2$
 $g_{12} = x_\xi x_\eta + y_\xi y_\eta$
 $g_{22} = x_\eta^2 + y_\eta^2$
 ω : vorticity of fluid
 ϕ : potential function
 D : divergence
 h : heaving displacement of flat plate
 h_0 : heaving amplitude of flat plate
 α : pitching angle of flat plate
 α_0 : pitching amplitude of flat plate
 Ω : angular frequency of oscillation of flat plate

- Φ : phase difference between heaving and pitching motion
 Γ_{bv}, Γ_{sv} : strengths of bound vortex and shed vortex
 M : number of bound vortices
 N : number of time steps in discrete vortex method
 u_∞ : velocity of uniform flow
 $z = x + iy$
 z_{bv}, z_{sv} : coordinates of bound vortex and shed vortex
 W : complex velocity potential
 F_x, F_y : x, y components of fluid force acting on body
 θ : feathering parameter
 C_T : thrust coefficient
 $\bar{P}$: mean thrust
 $\bar{E}$: mean energy consumption
 a : half-chord length
 η_e : propulsive efficiency
 $()_\xi, ()_\eta$: partial differentiation by ξ, η

2. Method of Analysis

2.1 Basic formulation

In two-dimensional incompressible viscous flow the Navier-Stokes equation and the equation of continuity are expressed in nondimensional form:

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{u} \quad (1)$$

$$\nabla \cdot \mathbf{u} = 0, \quad (2)$$

where $\mathbf{u} = (u, v)$. In order to solve these equations in curvilinear coordinates, we apply mapping from the physical plane (x - y) to the transformed plane (ξ - η) and solve these equations in the transformed plane, as shown in Fig. 1. The Navier-Stokes equation with respect to U and the equation of continuity on the transformed plane are expressed in the general tensor form⁽¹⁰⁾ as

$$\text{inertia term} : \frac{\partial U}{\partial t}$$

$$\text{convective term} : + \frac{1}{J} [(JU)_{\xi\xi} + (JU)_{\eta\eta}]$$

$$+ \frac{1}{J} [y_\eta (x_{\xi\xi} UU + 2x_{\xi\eta} UV + x_{\eta\eta} VV) - x_\eta (y_{\xi\xi} UU + 2y_{\xi\eta} UV + y_{\eta\eta} VV)]$$

Fig. 1 Mapping from physical plane (x - y) to transformed plane (ξ - η)

$$\text{pressure term} : = -\frac{1}{J^2} (g_{22} p_\xi - g_{12} p_\eta)$$

$$\begin{aligned} \text{viscous term} : & + \frac{1}{J} [(J\tau^{11})_\xi + (J\tau^{12})_\eta] \\ & + \frac{1}{J} [y_\eta (x_{\xi\xi}\tau^{11} + 2x_{\xi\eta}\tau^{12} + x_{\eta\eta}\tau^{22}) \\ & - x_\eta (y_{\xi\xi}\tau^{11} + 2y_{\xi\eta}\tau^{12} + y_{\eta\eta}\tau^{22})] \\ & - \frac{1}{J} \{ (JU)_\xi + (JV)_\eta \} = 0, \end{aligned} \quad (3)$$

where

$$\begin{aligned} \tau^{11} &= \frac{1}{JRe} \left\{ \frac{g_{22}}{J} U_\xi - \frac{2g_{12}}{J} U_\eta - \left(\frac{g_{22}}{J} \right)_\xi U - \left(\frac{g_{22}}{J} \right)_\eta V \right\} \\ \tau^{22} &= \frac{1}{JRe} \left\{ \frac{g_{11}}{J} V_\eta - \frac{2g_{12}}{J} V_\xi - \left(\frac{g_{11}}{J} \right)_\eta V - \left(\frac{g_{11}}{J} \right)_\xi U \right\} \\ \tau^{12} &= \frac{1}{JRe} \left\{ \frac{g_{11}}{J} U_\eta + \frac{g_{22}}{J} V_\xi + \left(\frac{g_{12}}{J} \right)_\xi U + \left(\frac{g_{12}}{J} \right)_\eta V \right\}, \end{aligned} \quad (4)$$

and U and V are defined as

$$\begin{bmatrix} U \\ V \end{bmatrix} = \begin{bmatrix} \xi_x & \xi_y \\ \eta_x & \eta_y \end{bmatrix} \begin{bmatrix} u \\ v \end{bmatrix}. \quad (5)$$

2.2 Discretization and method of solving pressure field

Although Eqs. (3) and (4) seem to be complicated, it is possible to discretize them similarly as in the SMAC method if JU and JV are used as variables to be solved. Figure 2 shows the placement of variables. The first-order forward-difference scheme for the inertia term and the third-order upstream-difference scheme for the convective term are employed. In the SMAC method, the tentative velocity of the fluid at each grid point is determined from the forward-difference of the Navier-Stokes equation using the old time step pressure, and the new time step pressure is determined so that the new time step velocity satisfies the equation of continuity. At this point, the problem is expressed as linear algebraic equations, in which the number of variables is equal to that of grid points. It is important to solve these linear equations efficiently in order to reduce the amount of calculation; many methods for solving these equations have been developed. In this paper we introduce Takemitsu's scheme⁽¹¹⁾ which is similar to the SMAC method but requires a much smaller number of iterations. Details of this scheme are not given in this paper, but an outline for the case of a uniform grid is

Fig. 2 Placement of variables

explained as follows.

First, vorticity ω is introduced as

$$\omega = \nabla \times \mathbf{u}. \quad (6)$$

Next, ω is assumed to be constant in each iteration, i.e.,

$$\omega^{(1)} = \omega^{(2)} = \dots = \omega^{(k)} = \omega^{(k+1)} = \dots, \quad (7)$$

where (k) denotes the number of iterations. Equation (8) is rewritten as

$$\omega = \nabla \times \mathbf{u}^{(k)} = \nabla \times \mathbf{u}^{(k+1)}, \quad (8)$$

from which we have

$$\mathbf{u}^{(k+1)} = \mathbf{u}^{(k)} - \nabla \phi^{(k+1)}. \quad (9)$$

If we take the divergence of Eq. (10),

$$D^{(k+1)} = D^{(k)} - \nabla^2 \phi^{(k+1)} \quad (10)$$

is obtained. If we assume that $D^{(k+1)} = 0$, we obtain

$$D^{(k)} = \nabla^2 \phi^{(k+1)}. \quad (11)$$

With use of Eq. (12), the potential function ϕ is calculated in each iteration, and the pressure p is corrected as

$$p^{(k+1)} = p^{(k)} + \frac{\phi^{(k+1)}}{\Delta t}. \quad (12)$$

Ultimately, Eq. (13) becomes equivalent to solving the divergence D for each iteration in the case of the SMAC method, but its overrelaxation is more effective than that of the SMAC method. We note that Eq. (12) is calculated iteratively using the Tschebyscheff SLOR method, with consideration of the vectorization for the supercomputer.

2.3 Extension to moving coordinate grid

The velocity of the coordinate grid can be calculated from the difference between the grid at the time step (n) and that at the time step $(n+1)$, i.e.,

$$\begin{aligned} JU_{\text{mesh}} &= y_\eta (x_{(n+1)} - x_{(n)}) / \Delta t - x_\eta (y_{(n+1)} - y_{(n)}) / \Delta t \\ JV_{\text{mesh}} &= x_\xi (y_{(n+1)} - y_{(n)}) / \Delta t - y_\xi (x_{(n+1)} - x_{(n)}) / \Delta t. \end{aligned} \quad (13)$$

As shown in Fig. 3(a), the velocity of the fluid at the time step $(n+1)$ can be obtained by replacing the velocity in the convective term of Eq. (3) with velocity relative to the moving grid.

$$\begin{aligned} & \frac{1}{J} [(JU)(JU - JU_{\text{mesh}}) / J]_\xi \\ & + \{(JU)(JV - JV_{\text{mesh}}) / J\}_\eta \\ & + U \{ (JU_{\text{mesh}})_\xi + (JV_{\text{mesh}})_\eta \} \\ & + \frac{1}{J^3} [y_\eta \{ x_{\xi\xi} (JU)(JU - JU_{\text{mesh}}) \\ & + x_{\xi\eta} (JU - JU_{\text{mesh}})(JV) + x_{\eta\eta} (JU)(JV - JV_{\text{mesh}}) \\ & + x_{\eta\eta} (JV - JV_{\text{mesh}})(JV) \} \\ & - x_\eta \{ y_{\xi\xi} (JU)(JU - JU_{\text{mesh}}) \\ & + y_{\xi\eta} (JU - JU_{\text{mesh}})(JV) + y_{\eta\eta} (JU)(JV - JV_{\text{mesh}}) \\ & + y_{\eta\eta} (JV - JV_{\text{mesh}})(JV) \}] \end{aligned} \quad (14)$$

With Eq. (15), U^* and V^* are obtained as

$$U^* = U + \Delta t \{ -(\text{convective term of } U)$$

$$+ (\text{pressure term of } U)$$

$$+ (\text{viscous term of } U) \}$$

$$V^* = V + \Delta t \{ -(\text{convective term of } V)$$

$$+ (\text{pressure term of } V) \\ + (\text{viscous term of } V) \}. \quad (16)$$

Next, as shown in Fig. 3(b), U^* and V^* , which are observed using the coordinate grid of the old time step, must be reobserved using the grid of the new time step. This reobservation (rezoning) is carried out using the definition of contravariant velocity:

$$\begin{aligned} (JU)_{n+1} &= y_{\eta(n+1)}(x_{\xi}U^* + x_{\eta}V^*) \\ &\quad - x_{\eta(n+1)}(y_{\xi}U^* + y_{\eta}V^*) \\ (JV)_{n+1} &= y_{\xi(n+1)}(x_{\xi}U^* + x_{\eta}V^*) \\ &\quad - x_{\xi(n+1)}(y_{\xi}U^* + y_{\eta}V^*). \end{aligned} \quad (17)$$

2.4 Procedure of present scheme

The coordinate grid should be corrected at each time step. This correction also requires iterative calculation, so that the total amount of calculation will increase enormously if the iterative calculation for the grid is performed in each time step. Therefore, in the present scheme, a set of grids for one oscillation cycle is calculated at the beginning of the entire calculation, and these grids are saved in the memory of a computer in order that the grids for one cycle can be read from the memory in each time step with no iterative calculation.

The procedure of the present scheme is summarized as follows.

- (1) Calculate the coordinate grids for one oscillation cycle.

Fig. 3 Extension to moving coordinate grid

Fig. 4 Analytical model

tion cycle.

- (2) Calculate U^* and V^* using Eq. (16).
- (3) Reobserve using new time step coordinate grid.
- (4) Correct the pressure field through iteration.
- (5) Advance the time step and return to (2).

3. Numerical Calculation

3.1 Analytical model

In order to verify the present scheme explained in the previous section, we calculate two-dimensional flow around an oscillating flat plate using the present scheme and the discrete vortex method, and compare the solutions from these two methods. Figure 4 shows the analytical model. The oscillation of the flat plate has two degrees of freedom, heaving and pitching motion.

$$\begin{aligned} h &= -h_0 \sin(\Omega t) \\ \alpha &= \alpha_0 \sin(\Omega t + \Phi) \end{aligned} \quad (18)$$

The ratio of amplitudes of heaving and pitching motion is expressed as the feathering parameter θ .

$$\theta = \frac{\alpha_0 u_\infty}{h_0 \Omega} \quad (19)$$

Figure 5 depicts the analytical field. The fluid flows in uniformly from the left and flows out freely toward the right. The top and bottom boundaries are assumed to be slip walls. Figure 6 shows the coordinate grids. The grids are divided finely near the flat plate and the number of divisions is (80×40) . Table 1 shows specifications of the calculation.

3.2 Analysis by the discrete vortex method

In the discrete vortex method, the boundary conditions for the body are represented by M bound vortices (their strengths are $\Gamma_{bv(j)}$), and the shear layer from the trailing edge is represented by shed vortices (their strengths are $\Gamma_{sv(k)}$). Figure 7 shows the placement of the vortices. Moreover, we assume that

- (1) the strengths of shed vortices are determined by Kelvin's theorem;
- (2) the velocities of shed vortices are equal to that of uniform flow;
- (3) the strengths of shed vortices do not decrease.

The respective strengths of bound vortices are

Fig. 5 Analytical field

Table 1 Specifications of calculation

time step size	0.002 (const.)
Reynolds number	16000
chord length	0.4
amplitude of heaving motion	0.08
cycle of oscillation $2\pi/\omega$	1 (= 500 step)
phase difference	$\pi/2$

(a) $t = 0$ (b) $t = 0.25$

Fig. 6 Coordinate grids

Fig. 7 Placement of vortices

determined such that the velocities normal to the flat plate at all control points are equal to the velocity of the wing. This problem becomes that of solving a set of M linear algebraic equations. Moreover, Kelvin's theorem is added as

$$\sum_{j=1}^M \Delta \Gamma_{bv(j)} + \Gamma_{sv(N)} = 0. \quad (20)$$

By solving these $(M+1)$ linear algebraic equations, the complex velocity potential of the flow at time step (N) becomes

(a) without rezoning

(b) with rezoning

Fig. 8 Comparison of results without rezoning and those with rezoning

$$W = u_{\infty} z + \frac{i}{2\pi} \sum_{j=1}^M \Gamma_{bv(j)} \log(z - z_{bv(j)}) + \frac{i}{2\pi} \sum_{k=1}^N \Gamma_{sv(k)} \log(z - z_{sv(k)}), \quad (21)$$

from which the complex velocity is obtained as

$$u + iv = \frac{dW}{dz}. \quad (22)$$

The fluid force is obtained from the generalized Blasius formula

$$F_x - iF_y = \frac{i\rho}{2} \oint_C \left(\frac{dW}{dz} \right)^2 dz + i\rho \frac{\partial}{\partial t} \oint_C \bar{W} dz, \quad (23)$$

where C represents an arbitrary closed curve surrounding the body.

3.3 Results and discussion

In order to verify the effect of rezoning, we first calculate uniform flow without the oscillating flat plate, using the oscillating coordinate grid shown in Fig. 6. In this case, if the oscillating grid does not affect the flow, the flow will be completely uniform. The results are shown in Fig. 8. In Fig. 8(a) the procedure of rezoning is omitted, so that the flow is affected by the grid and undulates. On the other hand, the results with rezoning are shown in Fig. 8(b). The flow becomes almost completely uniform. Therefore, it is considered that rezoning is a very important procedure.

In the case of flow with the flat plate, the streamlines and the pressure contours are shown in Fig. 9 and Fig. 10, respectively. The streamlines appear to run through the flat plate. The reason for this result is that the flat plate moves with the fluid. In Fig. 10 it can be observed that the vortices are shedded from the

trailing edge.

Next, in order to verify the accuracy of the present scheme, we regard the two-dimensional flat plate as a propulsion mechanism, and investigate the characteristics of its thrust and propulsive efficiency. In the present scheme the thrust is calculated as the x -component of the force F_p which is perpendicular to the flat plate, i.e.,

$$F_x = F_p \sin \alpha. \quad (24)$$

F_p is obtained by integrating the pressure difference between upper and lower sides of the flat plate. On the

(a) Entire field of analysis

(b) Field near the flat plate

Fig. 9 Streamlines ($Re=16\,000$, $t=2$)

Fig. 10 Pressure contours ($Re=16\,000$, $t=2$)

Fig. 11 Change of the suction force direction due to separation at the leading edge

other hand, in the discrete vortex method, the force acting on the flat plate is calculated using Eq. (23), so that a component parallel to the flat plate appears (Fig. 11(a)). This component of force, which is called leading edge suction force F_s , occurs due to high velocity (=low pressure) at the leading edge. In this paper, however, similarly to the calculation of the nonlinear lift force due to a separated vortex, we assume that the separated vortex appears on the top side of the flat plate and that the direction of the leading edge suction force changes from forward to upward. It is also considered that F_s does not contribute to the thrust, since F_s is relatively small compared to $|F_p|$ (Fig. 11(b)). Therefore, it is considered that the solutions of the present scheme correspond to those of the discrete vortex method in which the contribution of F_s is omitted. Figure 12 shows the relationship between θ and the thrust coefficient C_T , which is given by

$$C_T = \bar{P} / \Omega^2 h_0^2 a. \quad (25)$$

Figure 13 shows the relationship between θ and the propulsive efficiency η_e , defined by

Fig. 12 Relationship between C_T and θ

Fig. 13 Relationship between η_e and θ

$$\eta_e = \frac{\bar{P}u_\infty}{\bar{E}} \times 100 [\%], \quad (26)$$

where $\bar{E}$ denotes the mean energy consumption. In Figs. 12 and 13, DVM 1 and DVM 2 respectively represent the results with leading edge suction and those without it. The results obtained from the present scheme (FDM in Figs. 12 and 13) show good agreement with those obtained from the discrete vortex method without the leading edge suction force.

The ETA¹⁰ supercomputer at the Tokyo Institute of Technology was used for calculation. The calculating time is about one-half hour for the calculation to $t=8$ (time step 4 000).

4. Conclusions

A finite difference scheme for solving incompressible viscous flow around an oscillating body was presented. This scheme is an extension of the SMAC method to the moving coordinate grid and can be applied to flow around an arbitrary-shaped oscillating body. This scheme was applied to flow around an oscillating flat plate. We investigated the characteristics of its thrust and propulsive efficiency and examined the accuracy of the present scheme through comparison with the results obtained from the discrete vortex method. The results obtained from the present scheme show good agreement with those obtained from the discrete vortex method, if we neglect the contribution of the leading edge suction force.

References

- (1) Lighthill, M.J., Aquatic Animal Propulsion of High Hydromechanical Efficiency, *J. Fluid Mech.*, Vol. 44, No. 2 (1970), p. 265.
- (2) Kobayashi, H., Wada, A. and Kadoyu, M., Computational Fluid Experiment, (in Japanese), (1989), p. 74, Asakura-shoten.
- (3) Sakata, H., Adachi, T. and Inamuro, T., A Numerical Analysis of Unsteady Separated Flow by Vortex Shedding Model (1st Report, Flow around a Square Cylinder), *Trans. Jpn. Soc. Mech. Eng.*, (in Japanese), Vol. 49, No. 440, B (1983), p. 801.
- (4) Ogawa, T., A Study for Savonius Rotor (1st Report, Theoretical Analysis), *Trans. Jpn. Soc. Mech. Eng.*, (in Japanese), Vol. 49, No. 441, B (1983), p. 976.
- (5) Tsutahara, M., Kimura, T. and Ro, K., A Ship's Propulsion Mechanism of Two-Stage Weis-Fogh Type (Model and Analysis of Its Properties by a Singularity Method), *Trans. Jpn. Soc. Mech. Eng.*, (in Japanese), Vol. 53, No. 496, B (1987), p. 3711.
- (6) Harlow, F.H. and Welch, J.E., Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface, *Phys. Fluids*, Vol. 8, No. 12 (1965), p. 2182.
- (7) Amsden, A.A. and Harlow, F.H., A Simplified MAC Technique for Incompressible Fluid Flow Calculations, *J. Comput. Phys.*, Vol. 6, No. 2 (1970), p. 322.
- (8) Shin, R.B., Ikohagi, T. and Daiguji, H., An Efficient Finite-Difference Scheme for the Steady Incompressible Navier-Stokes Equations Using a Curvilinear Coordinate Grid, *Trans. Jpn. Soc. Mech. Eng.*, (in Japanese), Vol. 56, No. 525, B (1990), p. 1357.
- (9) Azuma, A., *Aeronautics (I)*, (in Japanese), (1989), p. 100, Shokabou.
- (10) Di Carlo, A., Piva, R. and Guj, G., A Study on Curvilinear Coordinate and Macro-Elements for Multiply Connected Flow Fields, *Lecture Notes in Phys.*, Vol. 90 (1979), p. 161, Springer-Verlag.
- (11) Takemitsu, N., Finite Difference Method to Solve Incompressible Fluid Flow, *J. Comput. Phys.*, Vol. 61 (1985), p. 499.