A Finite-Difference Method for a Separating Flow Past a Body of Arbitrary Geometry in Rectangular Coordinate Systems

by Yoshihiro Yamada*, *Member*
Hideaki Miyata**, *Member*

Summary

A finite-difference method for a separating flow past a 3D body of arbitrary geometry is developed in the framework of rectangular coordinate systems. No-slip body boundary conditions are approximately implemented in the boundary cells making use of two kinds of porosity parameters. The availability of the method is demonstrated by the simulation of a flow past a sphere at the Reynolds number 1000. The complete 3D vortex shedding motion is well simulated.

1. Introduction

A variety of methods of computational fluid dynamics have been developed mostly in the last decade. Recent tendency is to shift the purpose of computation to the more applicational direction. Most of the methods employ the curvilinear coordinate system fitted to the body surface because the resolution of the boundary layer on the body surface is supposed to be of substantial importance. However, the generation of a 3D curvilinear coordinate system is difficult and time consuming, since the appropriateness of the coordinate system is certified only when the computation is succeeded by avoiding the excessive violation of the conservation laws due to the irregular grid system. Furthermore, it may be almost impossible to generate a smooth grid system for a body of complex geometry, such as those of an automobile with wheels or without a roof and a body beneath the breaking wave surface.

At the authors' laboratory a series of finite-difference method using a fixed rectangular coordinate system are developed since 1979. The TUMMAC-IV¹⁾²⁾ method for ship waves, the TUMMAC-V³⁾⁴⁾⁵⁾ method for general 2D flows and the TUMMAC-VI⁶⁾ method for a two-layer flow are the typical examples. The TUMMAC-IV method is now used as a design tool by five major shipbuilding companies in Japan. One of the major reasons of the practical usefulness of the TUMMAC-IV method is that the required effort for the grid genera-

tion is negligibly small due to the use of the rectangular coordinate system.

This paper describes the new TUMMAC method named TUMMAC-VII for a 3D separating flow past a body of arbitrary geometry. Following the above-mentioned methods the fixed rectangular coordinate system is employed and the no-slip body boundary conditions are implemented by newly-developed techniques. Details of the method are described and it is applied to a flow past a sphere, for which only very limited knowledge is so far obtained.

2. Computational method

2.1 Grid system and zoning

A rectangular staggered mesh system shown in Fig. 1 is employed. Since it is not fitted to the body surface, the degree of accuracy is not high for the implementation of the body boundary conditions and for the resolution of the physical phenomena in the vicinity of the body

Fig. 1 Velocity and pressure points.

* Graduate School, Department of Naval Architecture and Ocean Engineering, University of Tokyo.

** Department of Naval Architecture and Ocean Engineering, University of Tokyo

surface. However, this system has the advantages of high adaptability and high robustness. Furthermore, the elaborate efforts of grid generation are almost entirely eliminated, which is most important for the engineering applications. For the resolution of the separated flow far behind a body the equally-spaced grid system gives better resolution than the boundary-fitted coordinate system which provides coarser grid spacing in the far-field. As described in the subsequent sections, most of the efforts are focused on the treatments in the body boundary cells.

In order to increase the computational efficiency the computational domain is divided into the region of the finer grid system that includes the body and that of the coarser grid system that surrounds it. The former region is further divided into two regions, that is, the fluid region composed of full-of-fluid cells (F-cells) and the body boundary region composed of boundary cells (B-cells), as illustrated in Fig. 2.

2.2 Governing equations

The governing equations are the continuity equation and the Navier-Stokes equation as follows.

$$\partial_t u^i = 0, \quad (1)$$

$$\partial_t u^i = -\partial_i P + a^i. \quad (2)$$

Here, ∂_i denotes space-differencing in the x_i direction and ∂_t time-differencing. The last term a_i is written as

$$a^i = -u^j \partial_j u^i + (Re^{-1} + \nu_s) \partial_j \partial_j u^i + M^i + \partial_j \nu_s (\partial_j u^i + \partial_i u^j) + f^i, \quad (3)$$

where ν_s is the eddy viscosity coefficient of the SGS turbulence model, M^i is the controllable numerical dissipation introduced to stabilize the solution, and f^i is the external force.

2.3 Computational procedure

The algorithm of the computation is same with the previous TUMMAC method. Updating of the velocity field is made after updating the pressure field by solving the Poisson equation and this cycle is repeated in the time-marching procedure. Since a zonal method is employed, the computation is proceeded from the outer region with a coarse grid system to the inner region with a fine grid system. For the variables at the inter-

Fig. 2 Schematic sketch of zoning.

Fig. 3 Computational procedure.

face between two regions and between the regions of F-cells and B-cells the Dirichlet conditions are respectively imposed.

For the solution of the Poisson equation for the pressure the following Richardson method is used in the outer region as well as the inner region with F-cells.

$$\begin{aligned} \partial_i \partial_i P &= \partial_i (a^i + u^i / \Delta t) \\ &= \partial_i b^i, \end{aligned} \quad (4)$$

$$P^{m+1} = P^m + \omega (\partial_i \partial_i P - \partial_i b^i), \quad (5)$$

where ω is the relaxation factor and b^i is the source term. In the region composed of B-cells the following simultaneous iterative method is used, because it is very suitable for the implementation of the zero-divergence condition in B-cells.

$$P^{m+1} = P^m + \frac{\omega}{A} D, \quad (6)$$

$$A = \Delta t * \left[\frac{\gamma_{i+1} + \gamma_{i-1}}{(\Delta x)^2} + \frac{\gamma_{j+1} + \gamma_{j-1}}{(\Delta y)^2} + \frac{\gamma_{k+1} + \gamma_{k-1}}{(\Delta z)^2} \right], \quad (7)$$

where D is the divergence of a cell and γ is the volume-porosity defined in the subsequent session.

The velocity field is updated by the following equation.

$$u^{i(n+1)} = u^{i(n)} + \Delta t (-\partial_i P + a^i). \quad (8)$$

3. Body boundary condition

3.1 Porosity

In order to represent the complicated configuration of a 3D body two kinds of porosity are introduced, that is, volume-porosity (γ) and surface-porosity (β). The volume-porosity is the ratio of the fluid portion of each boundary cell and the surface-porosity is the ratio of the fluid portion in each surface of the boundary cell and estimated as shown in Table 1. Since the location of the body boundary is approximately represented by these scalar values, they are used not only for the flagging of cells and for the drawing of the body configuration but

Table 1 Definition of porosity for body boundary.

	volume porosity	surface-porosity
variable name	γ	β
definition	ratio of fluid volume in one cell	$\frac{\gamma + \gamma_{i+1}}{2}$
definition point	pressure point	velocity point
usage	flagging, choice of difference scheme	choice of difference scheme, divergence calculation, representation of body configuration

Fig. 4 Definition of body configuration by surface-porosity.

Fig. 5 Reproduced sphere in case $\Delta=1/20$.

also for the flux and divergence calculation and for the choice of differencing scheme.

In Fig. 4 the definition of a body configuration by use of surface-porosity is sketched. A cross-section of a sphere which is reproduced by this procedure is shown in Fig. 5.

3.2 Flagging

The boundary-cell (B-cell) is defined as a cell of which volume-porosity is greater than 0.5 but less than 1.0. A cell of which volume-porosity is less than 0.5 is defined as an empty cell (E-cell) where the pressure is not computed. A full-of-fluid cell (F-cell) facing a E-cell is defined as a special B-cell (B*-cell) with the porosity

Fig. 6 Cell flagging.

of 1.0. The choice of the computational method for the pressure by Eq. (5) or (6) is determined by this flagging.

The velocity point located at the center of each surface of a cell is assumed to be present in case the surface-porosity is greater than 0.5. The choice of the differencing scheme is also dependent on the surface-porosity.

3.3 Velocity extrapolation

The velocity on a cell surface of which surface-porosity is unity is normally calculated by the momentum equation (8), while that on a cell surface of which surface-porosity is greater than 0.5 but less than 1.0 is extrapolated from the neighboring velocities. The velocities which are not extrapolated themselves are used for the extrapolation and the surface-porosity of the cell where neighboring velocities are situated is used as a weighting function in the extrapolation by a quadratic equation. Therefore, the extrapolation is made approximately in the direction normal to the body surface.

3.4 Differencing scheme

By use of surface-porosity the differencing scheme is chosen since velocity does not exist on a cell-surface of which surface-porosity is less than 0.5. In the differencing scheme the no-slip condition is incorporated.

For the first derivative of velocity the fourth-order centered differencing is used. When available velocity points are not sufficient for this five-points scheme, the second-order centered differencing or the modified centered differencing is used considering the distance to the body surface and zero-velocity on the body surface.

For the second-derivative of velocity the second-order centered differencing is used and it is modified in the vicinity of the body surface considering the distance between the velocity point and the body surface as well as the zero-velocity.

Fig. 7 Velocity and pressure points at the interface.

Following the previous work⁷⁾ a controllable numerical dissipation term is added to keep the stability of the solution.

$$\begin{aligned}
 M^i = & -|u^j| \frac{\alpha_2}{2} \Delta^2 \partial_j^2 u^i \\
 & + u^j \frac{\alpha_3}{6} \Delta^2 \partial_j^3 u^i \\
 & - |u^j| \frac{\alpha_4}{24} \Delta^3 \partial_j^4 u^i.
 \end{aligned} \quad (9)$$

In the close vicinity of a body boundary where the five-points scheme for the third term of Eq. (9) cannot be used the parameters are set at $\alpha_2 = -1$, $\alpha_3 = 0$ and $\alpha_4 = 0$, while they are set at $\alpha_2 = \alpha_3 = 0$ and $\alpha_4 = 6.0$ elsewhere.

4. Zoning method

4.1 Zone division

The region with finer grid spacing is allotted so that it includes the body. The region with coarser grid spacing is continuously located. The coarser grid spacing is set at double of the finer one so that the grid lines of coarser system coincide with those of the finer system.

4.2 Interfacing

Two regions have an overlapping portion at the interface as shown in Fig. 7. The velocity and pressure in dotted marks are normally computed, while those in solid marks are given by linearly interpolating the values of the overlapped region. Since the computation is performed separately in each region as shown in Fig. 3, the use of interpolated values in the overlapped region means that the Dirichlet conditions are imposed at the interface.

5. Separating flow past a sphere

5.1 Condition of computation

A flow past a sphere is chosen for the computation to demonstrate the usefulness of the present method. It is partly because a sphere is one of the typical examples of complex geometry for rectangular coordinate systems and partly because the flow about a sphere is of significant importance from a scientific view-point. The

Table 2 Condition of computation.

	a	b	c
cell size = Δ	1/12	1/16	1/20
Domain of computation (fine) (coarse)	$9 \times 2 \times 2$ $9 \times 4 \times 4$	$12 \times 2 \times 2$ $12 \times 4 \times 4$	$9 \times 1.8 \times 1.8$ $9 \times 3.6 \times 3.6$
Number of used cell	9×10^4	26×10^4	30×10^4
Reynolds number	10^3	10^3	10^3
Time increment	1/120	1/160	1/200
Time steps for acceleration ($T = 5$)	600	800	1000
CPU - Time ($T = 20$) (TITAN - II)	30 h	100 h	150 h
Length and time are all made dimensionless with respect to the diameter of a sphere and the uniform velocity

Fig. 8 Computational domain.

Reynolds number based on the diameter of a sphere is set at 1000 throughout.

As listed in Table 2 the computation was performed with three grid systems to know the effect of grid spacing. The computation was continued until the dimensionless time T (with respect to the uniform flow velocity and the diameter of the sphere) reaches 20 to 30. The computational domain of the case in which the diameter of the sphere is divided into 16 cells ($\Delta = 1/16$, where Δ is the spacing) is longer than the other cases so that the motions of the separated vortices are simulated for a longer period (see Fig. 8).

All computations were performed by the graphic-supercomputer TITAN-II with two CPUs and 64MB core memory. The CPU time listed in Table 2 is only 3 to 5 times larger than the mainframe computer HITAC M-682H.

5.2 Computed results and discussion

The pressure distribution on the surface of a sphere at $T = 20$ is compared among three cases in Fig. 9. Since the spacing is not sufficiently fine and the body boundary conditions are very approximately implemented as described in the preceding section, the agreement with the averaged distribution cited from Ref. [8] is not satisfactory. However, overall agreement seems to be marginally good, considering that it is still very difficult to have satisfactory agreement even in the 2D case of a flow about a circular cylinder.

The pressure contours are shown both in the inner

Fig. 9 Pressure distribution on a sphere, (a) $\Delta=1/12$, (b) $\Delta=1/16$, (c) $\Delta=1/20$, vertical bar indicates the circumferential variation, the solid curve is from the experiment in Ref. [8].

region of fine spacing and in the whole region in Fig. 10. It is noted that the flow field is connected with sufficient smoothness across the interface.

The mechanism of separated flow past a sphere has been left vague. From the visualized flow field, such as one shown in Fig. 15, Taneda¹⁰⁾ elucidated that the most dominant vortices are the axisymmetric vortex ring and the tail vortices that emanate from the center of the former. Achenbach⁹⁾ tried to construct a simplified model of vortex structure. However, we do not yet have a clear idea of the 3D vortex structure about a sphere.

The simulated results are shown in Figs. 11 to 14, in which ω^1 is the vorticity of the streamwise vortex and ω^2, ω^3 those of the spanwise vortex. Another vorticity ω^r is specially defined in each cross-sections as

$$\omega^r = \omega^2 \sin \theta - \omega^3 \cos \theta, \quad (10)$$

where θ is the argument with the origin on the center-line. With this value the axisymmetric vortex ring can be revealed in the drawings. It is obviously noted that the vortex formation, which is observed by overlapping Fig. 11 on Fig. 12, has excellent resemblance with the observation in Fig. 15, although the Reynolds number is slightly different.

The simulated results explain the dominant features of the separated flow past a sphere as follows. (1) The axisymmetric spanwise vortex is elongated until its length reaches 2 to 3 times longer than the diameter. (2) in the core of the axisymmetric vortex secondary vortices are formed and they are seriously deformed to make unsymmetrical vortices, (3) from the core of the axisymmetric vortex tail-vortices, which are mostly composed of streamwise vorticity, are successively shed behind, and (4) it is periodically repeated at the Strouhal number of grossly 0.25. As observed in Figs. 13 and 14, the tail-vortices are usually made of one or two pairs of counter-rotating streamwise vortex tubes, and their periodical shedding is related with the periodical elongation of the axisymmetric vortex ring.

6. Concluding remarks

A new computational method TUMMAC-VII is developed for a 3D flow past a body of complex geometry in the framework of a rectangular coordinate system. The availability of the method is demonstrated by the simulation of a flow past a sphere, and the interesting phenomenon of vortex shedding which has been left obscure is elucidated. The present method is supposed to be very useful for the elucidation of the complicated 3D viscous flow about a complicated 3D geometry, such as the flow about an oil-drilling platform and the flow over arbitrary topography of the ocean floor.

Acknowledgements

This research is supported partly by the Grant-in-Aid for Cooperative Research of the Ministry of Education,

Fig. 10 Pressure contour maps on the $x-y$ central plane in the region of fine grid system (above) and in the whole domain (below), $\Delta=1/20$, $T=20$, the contour interval is 0.02, and the positive pressure is drawn in bold lines.

Fig. 11 Time-sequential development of ω^3 , $\Delta=1/16$, the contour interval is 0.5, and anticlockwise rotation is drawn in bold lines.

Science and Culture and partly by the LINEC group of shipbuilding companies. The authors are thankful to Professor H. Kajitani for his encouragement and to Miss. K. Onoda for her careful typewriting.

References

- 1) H. Miyata and S. Nishimura : Finite-difference simulation of nonlinear ship waves, J. Fluid Mech., vol. 157 (1985), pp. 327-357
- 2) H. Miyata, S. Nishimura and A. Masuko : Finite Difference Simulation of Nonlinear Waves Generated by Ships of Arbitrary Three-Dimensional Configuration, J. Comp. Phys., vol. 60 (1985), pp. 391-436
- 3) H. Miyata : Finite-Difference Simulation of

Fig. 12 Time-sequential development of ω^1 at the same instances with Fig. 11, the contour interval is 0.5, and the anticlockwise rotation observed from upstream is drawn in bold lines.

Fig. 13 Contours of ω^r and ω^1 on various cross-sections at $T=22.5$, $\Delta=1/16$, the contour interval is 0.5, and the anticlockwise rotation from upstream is drawn in bold lines.

Fig. 14 Same as Fig. 13 at $T=25.0$.

Fig. 15 Visualized flow-field at $Re=300$ from Taneda¹⁰⁾.

- 4) H. Miyata, H. Kajitani, M. Zhu, and T. Kawano: Nonlinear Forces Caused by Breaking Waves, Proc. 16th Symposium on Naval Hydrodynamics, Berkeley (July 1986)
- 5) H. Miyata et al.: Numerical and Experimental Analysis of Nonlinear Bow and Stern Waves of a Two-Dimensional Body (fourth report), J. Soc. Naval Archit., Jpn, vol. 157 (June 1985), pp. 141-154
- 6) H. Miyata, M. Katsumata, Y. G. Lee and H. Kajitani: A Finite-Difference Simulation Method for Strongly Interacting Two-Layer Flow, J. Soc. Naval Archit., Jpn, vol. 163 (June 1988), pp. 1-16
- 7) N. Baba, H. Miyata: Effect of the form of Navier-Stokes equation on large-scale vortex dynamics, Proc. 1st Symposium on Numerical Fluid Dynamics, Chuo-University, Tokyo, (1987), pp. 195-198
- 8) H. Schlichting: Boundary-Layer Theory, Seventh edition, McGraw-Hill, Inc. (1979), p. 21
- 9) E. Achenbach: Vortex shedding from sphere, J. Fluid Mech., vol. 62, part 2 (1974), pp. 209-221
- 10) S. Taneda: Fluid dynamics studied by photographic image, Asakura-Shoten, Jpn (1988), pp. 38-39