A Numerical Scheme with Unstructured Moving Grid System and Its Application to Two-Dimensional Complicated Flows

by Moon-Chan Kim*, *Member* Kazu-hiro Mori**, *Member*
 Yasuaki Doi**, *Member* Qi Xu**, *Member*

Summary

A finite volume method in unstructured grid system is developed for the study on complicated two-dimensional low Reynolds number flows. The numerical method is first validated by a cavity flow problem, and then applied to some problems such as a surface flow accelerator and a rotating body problem which would be difficult to solve by a structured grid.

A moving grid system is introduced and applied to an unsteady problem. The developed moving grid system makes it possible to change the grid fast to fit a moving body at every time step without any regeneration of grid. The present computations show the possibilities of applying the developed technique to the analysis of flows around multiple moving bodies.

1. Introduction

With the current rate of progress in computational fluid dynamics(CFD)¹⁾, as well as with grid generation techniques and the enhancements in computer capability, it is now practical to simulate complicated fluid dynamics phenomena associated with realistic geometry. Flows around moving bodies of complicated geometry are sophisticated and interesting. The studies on such flows are expected to be a basic research for the motions of new *micro-hydro-machine* which is very small robot working in highly viscous fluid. Such studies on the *micro-hydro-machine* can be applied to various engineering fields. Motivated by such needs, in the present study, a new numerical scheme to simulate an unsteady extremely low Reynolds number flow is established and an unstructured moving grid system is developed and applied to unsteady flows around a moving body.

The grid generation is an important issue for a practical application of computation fluid dynamics which involves complicated geometry and/or motions. As a structured grid system has its limitation for the problems of complicated geometry, several grid generation techniques have been proposed to overcome its limitation. The unstructured grid method is one of the most useful methods in grid generation because of its greater flexibility of geometry handling.

It is difficult to analyze an unsteady flow around a

moving body without any change of grid. One of the easiest ideas is the regeneration of grid at every time step, but this is too time-consuming and also takes too much time to interpolate in the whole domain because the number and the location of grid points change at every time step. The developed moving grid system makes it possible to change the grid fast to fit the body configuration at every time step without any regeneration of grid. The boundary division method, proposed by Cavendish²⁾ for finite element method, is modified to generate multi-connected moving grid. The present computations show the possibilities of applying the moving grid system to the flow simulation around multiple moving bodies.

For the formulation in the unstructured grid system, finite volume method is used for the incompressible Navier-Stokes equations. Artificial compressibility which was firstly proposed by Chorin³⁾, for a steady flow problem and successfully developed to unsteady problem by Soh⁴⁾, is introduced in the continuity equation. An artificial dissipation term⁵⁾ is also used to remove the numerical instability due to the decoupling of neighbor nodes. This method has been successfully applied to free surface flow problem by Hino⁶⁾. An unsteady analysis method is also developed for a moving grid system.

The cavity flow is first calculated to validate the developed code. The developed code is then applied to the flow around rotors of various shapes and to a surface flow accelerator. These problems would be difficult to solve by a structured grid. Finally, the analysis of an unsteady flow around an arbitrary moving body is carried out by the developed scheme.

* Graduate School, Hiroshima University

** Faculty of Engineering, Hiroshima University

2. Formulation for Unstructured Grid

2.1 Formulation of Steady Flow

Two-dimensional incompressible Navier-Stokes equations are employed as governing equations which are expressed in the non-dimensional conservative form in Cartesian coordinates (x, y) . Steady equations with artificial compressibility terms are as follows:

$$\frac{\partial p}{\partial \tau} + \beta^2 \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) = 0 \quad (1)$$

$$\frac{\partial u}{\partial \tau} + \frac{\partial}{\partial x}(u^2 + p) + \frac{\partial}{\partial y}(uv) = \frac{1}{R_n} \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \quad (2)$$

$$\frac{\partial v}{\partial \tau} + \frac{\partial}{\partial x}(uv) + \frac{\partial}{\partial y}(v^2 + p) = \frac{1}{R_n} \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) \quad (3)$$

where τ is pseudo-time, (u, v) are the velocity components and p is pressure. The variables are normalized by principal velocity U and length L . The Reynolds number R_n is defined as $\frac{UL}{\nu}$ where ν is kinematic viscosity. β^2 is an artificial compressibility parameter defined as

$$\beta^2 = \gamma_b \times \text{Max.}(u^2 + v^2, \beta_{min}^2)$$

where γ_b is a global constant and the parameter β_{min}^2 is used to prevent β^2 from approaching zero near the stagnation point. $\gamma_b = 5$ and $\beta_{min}^2 = 0.0001$ are used in the present computations.

When the solution reaches to be converged, namely $\frac{\partial p}{\partial \tau}$ becomes almost zero, the equation (1) is the same as the original continuity equation. Equations (1) ~ (3) can be rewritten in the vector form as

$$\frac{\partial \mathbf{q}}{\partial \tau} + \mathbf{P} \left(\frac{\partial \mathbf{f}}{\partial x} + \frac{\partial \mathbf{g}}{\partial y} \right) + \mathbf{Q} \left(\frac{\partial \mathbf{f}_v}{\partial x} + \frac{\partial \mathbf{g}_v}{\partial y} \right) = 0 \quad (4)$$

where,

$$\mathbf{q} = \begin{bmatrix} p \\ u \\ v \end{bmatrix}, \quad \mathbf{f} = \begin{bmatrix} u \\ u^2 + p \\ uv \end{bmatrix}, \quad \mathbf{g} = \begin{bmatrix} v \\ uv \\ v^2 + p \end{bmatrix}, \quad \mathbf{f}_v = \begin{bmatrix} 0 \\ \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial x} \end{bmatrix}, \quad \mathbf{g}_v = \begin{bmatrix} 0 \\ \frac{\partial u}{\partial y} \\ \frac{\partial v}{\partial y} \end{bmatrix}$$

$$\mathbf{P} = \begin{bmatrix} \beta^2 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}, \quad \mathbf{Q} = \begin{bmatrix} 0 & 0 & 0 \\ 0 & \frac{1}{R_n} & 0 \\ 0 & 0 & \frac{1}{R_n} \end{bmatrix}$$

2.2 Spatial Discretization

The fluid domain is discretized by triangular elements and the flow variables (u, v) and p are defined at the vertices of each triangle. The control volume for a given node i is taken as the union of all the triangles which share that node as a vertex and point k is defined as the center point of edges between point j and point $j+1$ as shown in Fig. 1.

The integration of the governing equation (4) over this control volume yields

$$\iint_{\Omega} \frac{\partial \mathbf{q}}{\partial \tau} dxdy = - \iint_{\Omega} \mathbf{P} \left(\frac{\partial \mathbf{f}}{\partial x} + \frac{\partial \mathbf{g}}{\partial y} \right) dxdy$$

$$- \iint_{\Omega} \mathbf{Q} \left(\frac{\partial \mathbf{f}_v}{\partial x} + \frac{\partial \mathbf{g}_v}{\partial y} \right) dxdy$$

$$= - \int_{\partial \Omega} \mathbf{P} (f dy - g dx)$$

Fig. 1 Geometric identification of the control volume.

$$- \int_{\partial \Omega} \mathbf{Q} (f_v dy - g_v dx) \quad (5)$$

where Ω is the control volume and $\partial \Omega$ is its boundary.

Even for a moving grid system, equation (5) is valid because the control volume is previously set up at every time step. Namely, the location of new grid points is previously determined for the computation of a moving grid system.

In the discrete form, equation (5) can be written as

$$\left(S_i \frac{d\mathbf{q}_i}{d\tau} \right) = \sum_{k=1}^{n_e} \mathbf{P} (f_k \Delta y_k - g_k \Delta x_k)$$

$$+ \sum_{k=1}^{n_e} \mathbf{Q} (f_{vk} \Delta y_k - g_{vk} \Delta x_k) \quad (6)$$

where,

$$\Delta x_k = x_{j+1} - x_j, \quad \Delta y_k = y_{j+1} - y_j,$$

S_i is the area of the control volume around the node i , n_e is the number of edges, (x_j, y_j) is the coordinate at the node j and f_k and g_k are the flux vector evaluated by taking average of the values at both ends of the edge. f_{vk} is computed by the following relation.

$$f_{vk} = \int_{A_k} f_v dxdy / A_k \quad (7)$$

where A_k is the area of the two triangles associated with point k as shown in Fig. 1. g_{vk} is evaluated in the same way. In the boundary region, image points are added to the interior of boundary for the computation of terms f_{vk} and g_{vk} .

As the present discretization corresponds to the central difference scheme in the structured grid case, it is not stable due to the decoupling of neighboring node unless an artificial dissipation term is added to the equation. To keep the second order accuracy of the scheme, a fourth-order dissipation term is used in the scheme, which has been successfully applied to the incompressible Navier-Stokes equations^{5,6)}. The dissipation term is constructed by using $\Delta^2 \mathbf{q}$ as

$$D_i(\mathbf{q}) = \sum_{j=1}^{n_n} \epsilon \lambda_{ij} (\Delta^2 \mathbf{q}_j - \Delta^2 \mathbf{q}_i) \quad (8)$$

where

$$\Delta^2 \mathbf{q}_i = \sum_{j=1}^{n_n} (\mathbf{q}_j - \mathbf{q}_i),$$

n_n is the number of the boundary nodes around the node

i , ε is a global constant which controls the amount of dissipation, $\varepsilon=0.01$ is used in the present computations and λ_{ij} is a scale factor of wave speed. From the analogy with upwind scheme, λ_{ij} is determined as

$$\lambda_{ij} = \text{Max}(|w_i|, |w_j|) + \sqrt{\text{Max}(c_i, c_j)} \quad (9)$$

where

$$w_i = u_i \Delta y_{ij} - v_i \Delta x_{ij}$$

$$c_i = w_i^2 + \beta_i^2 \sqrt{\Delta x_{ij}^2 + \Delta y_{ij}^2}$$

and $(\Delta x_{ij}, \Delta y_{ij})$ is vector from the node i to the node j and w_j and c_j can be evaluated in the same way.

The evaluated dissipation term $(D_i(q))$ is added to the equation (6).

2.3 Algorithm for Unsteady Flow Analysis

For the unsteady problem formulation, the physical time t should be introduced. The formulation for unsteady problem is shown in the discretized form as follows:

$$\frac{p_i^{n+1,m+1} - p_i^{n+1,m}}{\Delta \tau} + F(u_i^{n+1,m}, v_i^{n+1,m}) = 0 \quad (10)$$

$$\frac{u_i^{n+1,m+1} - u_i^{n+1,m}}{\Delta t} + \frac{u_i^{n+1,m+1} - u_i^{n+1,m}}{\Delta \tau} + G(u_i^{n+1,m}, v_i^{n+1,m}, p_i^{n+1,m}) = 0 \quad (11)$$

$$\frac{v_i^{n+1,m+1} - v_i^{n+1,m}}{\Delta t} + \frac{v_i^{n+1,m+1} - v_i^{n+1,m}}{\Delta \tau} + H(u_i^{n+1,m}, v_i^{n+1,m}, p_i^{n+1,m}) = 0 \quad (12)$$

where Δt and $\Delta \tau$ are the physical and pseudo-time increments, m is the step number for the pseudo-time (τ), n is for the physical time (t), and F , G , H are residual functions.

$$q^{n+1,1} = q^n \text{ at } \tau=0,$$

q^n is converged value at physical time step n .

For the analysis of moving problem, the computed variables of physical time step n should be renewed at the new location of time step $n+1$, because the control volume moves from the location of physical time step n to the location of physical time step $n+1$. As the computation of residual functions (F , G , H) is carried out at the location of time step $n+1$, only the variables of u_i^n , v_i^n which were computed at the location of physical time step n (present computing time step is

$n+1$), are renewed at the new location as follow:

$$q_i^n = q_i^* + \frac{\partial q_i^*}{\partial l} \cdot \Delta l \quad (13)$$

where $*$ denotes previous location, Δl is the distance vector from the location at time step n to the location at time step $n+1$.

3. Application to Steady Flow

3.1 Grid Generation

The code of grid generation is developed based on the boundary division algorithm²⁾ which is well suited to an arbitrary multi-connected planar domain. The scheme is capable of automatically generating grid of triangular elements for the decomposition of virtually any multi-connected planar domain. The desired density of elements in various regions of the problem domain is specified by the user.

Co-ordinates of boundary nodes, interior boundary nodes and nodal density information are input data for the code. The interior nodal points are first generated with the consistence of nodal density information. The generated interior nodal points are connected to form triangles with requirement that the triangles be as close to equilateral ones as possible, and finally, triangles are smoothed by an average scheme.

As shown in Fig. 2, for example, the grid is successfully generated by the developed code.

3.2 Validation

The well-known cavity flow problem is analyzed to validate the developed solver. To check grid dependency, three kinds of grid shown in Fig. 2 are adopted. The u -velocities computed by the three kinds of grid are compared with those of finite difference method⁷⁾ at $x=0.5$, $R_n=500$ as shown in Fig. 3. A 50×50 structured grid and a third-order upwind scheme are used in the finite difference method. As little difference is found between the results obtained by the two methods, the developed unstructured scheme is acceptable. The case of grid-C is closer to the finite difference case. This is due to that the fine grid in the boundary layer region of

a) grid-A (no. of elements=3299) b) grid-B (no. of elements=5666) c) grid-C (no. of elements=5250)

Fig. 2 Generated unstructured grid for the cavity flow.

Fig. 3 Comparison of the computed u -velocity at $x=0.5$, $R_n=500$.

grid-C can simulate the flow more accurately. The computed velocity vectors and pressure contours are shown in Fig. 4.

In explicit time marching scheme, the maximum permissible time step is limited by Courant-Friedrichs-Lewy (CFL) and diffusion conditions as follows:

$$\Delta t \leq CFL \frac{S_i}{\lambda_i} \quad (14)$$

$$\Delta t \leq D \cdot R_n \cdot (S_i)^2 \quad (15)$$

where CFL is the maximum permissible Courant number, D is the maximum permissible diffusion number, S_i is the area of control volume at vertex i and λ_i is the scale factor of wave speed. The maximum Δt is determined by equations (14) and (15). $CFL=1$ and $D=0.5$

Fig. 4 Computed results of the cavity flow with the grid-A at $R_n=500$.

are used in the present computations. The computational conditions are summarized in Table 1. The converging rate of solution is shown in Fig. 5 and the dependency on Δt is also checked and the result is shown in Table 2.

3.3 Computations of Some Steady Flows

Analysis for some low Reynolds number flows around bodies of complicated geometry is carried out. The study on such flows is necessary for the research on the *micro-hydro-machine*. The first one is the flow around a rotor whose configuration is a circle, a square or a

Table 2 Validation of the Δt dependency with comparisons of u -velocity at $x=0.5$, $R_n=500$.

y	$\Delta t = 0.001$ (reference value)	$\Delta t = 0.002$ $(\frac{\Delta t=0.002}{\Delta t=0.001} - 1)$	$\Delta t = 0.005$ $(\frac{\Delta t=0.005}{\Delta t=0.001} - 1)$
0.0	0.0000	-	-
0.1	-0.1064	2.0×10^{-4}	2.0×10^{-4}
0.2	-0.2493	0.8×10^{-4}	0.8×10^{-4}
0.3	-0.3340	0.7×10^{-4}	0.9×10^{-4}
0.4	-0.2667	0.8×10^{-4}	1.1×10^{-4}
0.5	-0.1204	2.2×10^{-4}	2.5×10^{-4}
0.6	0.0169	-1.1×10^{-4}	-1.1×10^{-4}
0.7	0.1379	-0.1×10^{-4}	-0.2×10^{-4}
0.8	0.2600	-0.1×10^{-4}	0.2×10^{-4}
0.9	0.3689	-0.1×10^{-4}	0.3×10^{-4}
1.0	1.0000	-	-

Table 1 Computational conditions.

Cases	R_n	length(L) and velocity(U) for the definition of R_n	Δt	T (total time)
Cavity flow (grid-A)	500	L : edge length U : u -velocity(at $y = 1.0$)	0.002	50
Rotating flow	8	L : diameter(rotor) U : $\omega \times$ radius(rotor)	0.01	50
Surface flow Accelerator	100	L : diameter(rotor) U : $\omega \times$ radius(rotor)	0.002	100
Arbitrary motion (Unsteady)	10	L : diagonal of square U : $v_{translating} + \omega \times \frac{L}{2}$	0.01 ($\Delta \tau$) ($\Delta t = 0.05$)	20(τ)

Fig. 5 Converging rate during the computation of cavity flow at $Re=500$. (Maximum value of the rate is adopted in the whole domain)

cross as shown in Fig. 6. The Reynolds number is 8 and the no-slip condition is imposed on the outer boundary. Zero-gradient condition is used as the pressure boundary condition. The computed velocity field is shown in Fig. 7. As expected, if the angular velocities of the rotors is the same ($\omega=1.0$), the flow around the rotor of cross shape are the most disturbed. This computation shows that the unstructured solver provides a powerful tool for analyzing the rotating flow around any complicated configuration.

The second example is a surface flow accelerator as shown in Fig. 8 which is commonly used in circulating water channels. A slip condition is imposed on the upper boundary behind of the accelerator and the total number of triangular elements is 8233. This problem would be difficult to solve by a single structured grid because of the small distance between the accelerator and the outer casing and the sharp corner of the casing. Moreover, unstructured grid scheme effectively controls the density of element by adapting it to the characteristics of the flow. The circumferential velocity on the surface of the accelerator ($\omega r, r=1.0$) is set to be four

Fig. 6 Generated grid for the rotating flows around three kinds of rotor.

Fig. 7 Computed velocity vectors of the rotating flow around three kinds of rotor.

Fig. 8 Generated grid for the surface flow accelerator.

Fig. 9 Close-up view of the computed velocity vectors around the accelerator at $R_n=100$.

Fig. 10 Close-up view of the interpolated velocity vectors near the surface.

times the oncoming flow. Fig. 9 shows the computed velocity vectors around the accelerator, where a little reverse flow is found near the outer casing. Fig. 10

shows the interpolated velocity fields, from which it can be seen that the flow behind the rotor is well accelerated to reach a uniform flow. The above computation is another argument that steady highly viscous flows around bodies of complicated geometry can be simulated without any difficulties by the unstructured solver.

4. Moving Grid System for Unsteady Motion

4.1 Moving Grid System

A moving grid system is introduced for the computation of the unsteady flow around arbitrary moving bodies as illustrated in Fig. 11. The new grid system consists of two zones; one is a background zone and the other is a movable zone which moves with the body. In the movable zone, there is not only the zone of translation but also the zone of rotation as shown in Fig. 12. Both zones of translation and rotation are respectively divided into two zones, one is a buffer zone and the other is a translating or a rotating zone as shown in Fig. 12. The translating buffer zone shrinks or stretches to connect the translating zone to the background zone. The rotating zone can be smoothly connected with the irrotational zone through the rotating buffer zone. With the combined function of the translation and rotation, a body can arbitrarily move. Whenever reconnection is carried out, renumbering for connectivity information is also done in the moving grid system.

The developed moving grid system is highly time-saving because only connecting and renumbering are necessary at every time step, instead of the regeneration of the whole grid.

4.2 Computation of the flow around an arbitrary moving square

The developed moving grid system is applied to the flow around a square which undergoes prescribed translating and rotating motion. In the present computation, only a single body is considered, although there is no limitation in the present scheme to compute the motions of multiple bodies or a passive motion. The trajectory of the moving square is shown in Fig. 13 where t means the non-dimensional time. The moving square is assumed to be translating and rotating simultaneously as shown in the following equations.

$$u = u_{\text{translating}} + u_{\text{rotating}} \quad (16)$$

$$v = v_{\text{translating}} + v_{\text{rotating}} \quad (17)$$

where,

$$u_{\text{translating}} = A_x \left(\sin \left(0.4\pi \times \frac{t}{T} \right) - 0.2 \right)$$

$$v_{\text{translating}} = A_y \left(\cos \left(0.5\pi \times \frac{t}{T} \right) - 0.2 \right)$$

$$(u_{\text{rotating}}, v_{\text{rotating}}) = \omega \times \mathbf{r},$$

A_x and A_y are amplitudes of velocity, T is the total time, ω is angular velocity and the direction of rotation is counter-clockwise. $A_x=0.15$, $A_y=0.105$, $T=27$ and $\omega=1.0$ are used in the present computation. In this computation, ω is assumed to be constant, but the present scheme has no limitation to compute arbitrary rotating motion.

Fig. 11 Illustration of the moving grid system.

Fig. 12 Illustration of the movable grid zone.

Fig. 13 Trajectory of the moving square.

The Reynolds number and the maximum velocity is about 10 and 0.3, respectively. The generated grid at $t=27$ is shown in Fig. 14. The number of triangular elements in the background zone is 2800, that of the movable zone is 1004. The background zone is a triangular structured grid. The no-slip condition is imposed on the outer boundary. Fig. 15 shows the computed velocity vectors at $t=0.25$ and $t=27$. The renumbering and the correction for the moving grid system seem to be well carried out. The close-up view more clearly shows the flow patterns around the moving square. The flow at $t=0.25$ is not fully disturbed by the moving square but the flow at $t=27$ is disturbed in the whole domain and the flow direction is along the trace of the moving square. The flow direction is clearly shown by the streamlines in Fig. 16. As shown in Fig. 16, the rotating effect of the moving square to the flow can be seen in the streamline pattern.

Fig. 14 Generated grid for the moving square at $t=27$. Fig. 16 Streamline profile of the flow around the moving square at $t=27$. Fig. 15 Computed velocity vectors of the flow around the moving square at $t=0.25$ and $t=27$, respectively.

The scheme developed in the present study is unfit to analyze the case of a body being close to another one or boundary because of using the buffer zones. This limitation can be removed by introducing more sophisticated algorithm to handle such situations. It can be concluded that the present scheme is useful to analyze practical and realistic highly viscous flows.

5. Concluding Remarks

An unstructured grid system for simulating low Reynolds number flows around bodies of complicated geometry has been developed. The developed method is successfully applied to the analysis of steady flows around the bodies of complicated geometry which would be difficult to solve by a structured grid.

A moving grid system is proposed for the analysis of unsteady flow around a moving body. The capability of the developed moving grid system has been demonstrated in the present paper. The proposed scheme can be extended to the analysis of a passive motion and motions of multiple bodies. Although the present numerical scheme still has a room for improvement, the scheme can be the powerful tool for the basic research such as the motion of the *micro-hydro-machine*.

References

- 1) T. J. Barth: Computational Fluid Dynamics: Aspect of Unstructured Grids and Finite-Volume Solvers for the Euler and Navier-Stokes Equations, Lecture Series 1994-05, von Karman Institute for Fluid Dynamics, (1994), pp. 1-140.
- 2) James C. Cavendish: Automatic Triangulation of Arbitrary Planar Domains for the Finite Element Method, International Journal for Numerical Method in Engineering, Vol. 8, (1974), pp. 679-696.
- 3) A. J. Chorin: A Numerical Method for Solving Incompressible Viscous Flow Problem, Journal of Computational Physics, Vol. 2, (1967), pp. 12-26.
- 4) W. Y. Soh: Unsteady Solution of Incompressible Navier-Stokes Equation, Journal of Computational Physics, Vol. 79, (1988), pp. 113-134.
- 5) A. Jameson, et al.: Finite Volume Solution of the Two-Dimensional Euler Equations on a Regular Triangular Mesh, AIAA Journal, Vol. 24, No. 4, (1986), pp. 611-618.
- 6) T. Hino: A Finite Volume Method with Unstructured Grid for Free Surface Flow Simulations, The Proceedings of Sixth International Conference on Numerical Ship Hydrodynamics,

- (1993), pp. 173-188.
- 7) R. K. Agarwal: A Third-Order-Accurate Upwind Scheme for Navier-Stokes Solution at High Reynolds Number, AIAA paper No. 81-0112, (1981).
-