韓國港灣學會 第13卷 第1號, pp.167～173. 1999.

論 文

Computation of Two-Fluid Flows with Submerged Hydrofoil by Interface Capturing Method

*S. H. Kwag**

접면포착법에 의한 수중익 주위의 이층류 유동계산

곽 승 현

Key Words : Finite Volume Method(유한체적법), Two Fluid Flow(2층류), Submerged Hydrofoil(수중날개) Free Surface Flows(자유표면류), Interface Capturing Scheme(접면포착법)

Abstract

Numerical analysis of two-fluid flows for both water and air is carried out. Free-Surface flows with an arbitrary deformation have been simulated around two dimensional submerged hydrofoil. The computation is performed using a finite volume method with unstructured meshes and an interface capturing scheme to determine the shape of the free surface. The method uses control volumes with an arbitrary number of faces and allows cell-wise local mesh refinement. The integration in space is of second order, based on midpoint rule integration and linear interpolation. The method is fully implicit and uses quadratic interpolation in time through three time levels. The linear equation systems are solved by conjugate gradient type solvers, and the non-linearity of equations is accounted for through Picard iterations. The solution method is of pressure-correction type and solves sequentially the linearized momentum equations, the continuity equation, the conservation equation of one species, and the equations for two turbulence quantities.

1. Introduction

When the hull form is relatively simple, the interface tracking approach is convenient in which only the water flow is computed and the grid moves to adapt to the free surface.

* 정회원, 한라대학교 조선공학과 교수

Computation of flows, including the deformation of the free surface, is one of the big challenges in ship hydrodynamics. Many methods[1),2)] of this kind have been developed and successfully applied to flows. However, if the hull form is complicated, these methods are difficult to use because the grid has to be adapted both to the free surface and hull shape; the grid may deform too much in this process and a re - griding may become necessary.

Another problem which is difficult to handle using interface tracking methods is the breaking overturning, or wave splashing. In many cases, it is important to compute flows of special region, interface capturing methods have to be used. The MAC[3)] and VOF[4)] methods are of this kind, although the air flow is not both liquid and gas simultaneously, especially when the gas is closed by liquid and buoyancy effects become important.

Recently the density function method[5)] and two-fluid models of Ubbink[6)] and Muzaferija and Peric[7)] have been developed. The last is used here to demonstrate its applicability to the submerged hydrofoil flows. It is based on the finite volume(FV) approach and uses unstructured grids with arbitrary polyhedral control volumes(CVs). Both air and water are considered as a single fluid with variable properties. An additional transport equation for a void fraction of liquid is solved to determine the interface between the two fluids. A special discretized scheme for convective fluxes in the void fraction equation is used to ensure the sharpness of the interface. The results are presented for the computation of flows around the NACA 0012 hydrofoil.

2. Basic Euations and Numerical Strategy

Basic equations for mass, momentum and volume fraction are expressed in their integral form, which for an arbitrary, moving control volume can be written as,

$$\frac{d}{dt}\int_V \rho dV + \int_S \rho(\boldsymbol{v}-\boldsymbol{v}_b)\cdot\boldsymbol{n}\cdot dS = 0 \quad (1)$$

$$\frac{d}{dt}\int_V \rho u_i dV + \int_S \rho u_i(\boldsymbol{v}-\boldsymbol{v}_b)\cdot\boldsymbol{n}\cdot dS = \int_S(\tau_{ij}-p i_i)\cdot\boldsymbol{n}\cdot dS + \int_V \rho b_i dV \quad (2)$$

$$\frac{d}{dt}\int_V c dV + \int_S c(\boldsymbol{v}-\boldsymbol{v}_b)\cdot\boldsymbol{n}\cdot dS = 0 \quad (3)$$

In these equations, c is the void fraction of one phase; c=1 for liquid, c=0 for air or gas, and ρ is the fluid density, V is the control volume (CV) bounded by a closed surface S, $\boldsymbol{v}$ is the fluid velocity vector whose Cartesian components are u_i, $\mathbf{v}_b$ is the velocity of the CV surface, t is time, p is the pressure, b_i is the body force in the direction of the Cartesian coordinate x_i, $\boldsymbol{n}$ is the unit normal to S and directed outwards, and τ_{ij} are the components of the viscous stress tensor defined for (Newtonian incompressible fluids considered here) as,

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \quad (4)$$

with μ being the dynamic viscosity of the fluid. When the control volume moves, the so called *space conservation law*(SCL) has also to be satisfied; it is expressed by the following relation between the rate of change of CV and its surface velocity:

$$\frac{d}{dt}\int_V dV - \int_S \boldsymbol{v}_b\cdot\boldsymbol{n}\cdot dS = 0 \quad (5)$$

These equations are applied to each CV and discretized in order to obtain one algebraic equation per CV; each equation involves the unknown from the CV-center (where all knowns are stored) and from a certain number of neighbor CVs. Here second-order approximations (linear interpolation, central differences, and midpoint rule integration) are used to evaluate the integrals in space and time. The method is fully implicit, i.e., the spatial integrals are evaluated at the new time level, while the old values appear only in the approximation of the time derivative (linear or quadratic backward scheme). For more details on discretization methods, see[6),7)]. Here only the interface capturing features of the method will be briefly discussed.

The time involves three time levels and performs the integration over an interval centered around the new time level t_m. i.e., $t_m - \delta t_m/2$ to $t_m + \delta t_m/2$. The values at t_m, the center of the integration interval, represent the mean values for the integration period, which is a second- order approximation. A second-order approximation assumes a quadratic profile in time; when fit to values at three time levels, it leads to a second-order approximation of the derivatives at t_m.

Both fluids are treated as a single effective fluid whose properties vary in space according to the volume fraction of each phase. i.e.,

$$\rho = \rho_1 c + \rho_2 (1-c), \quad \mu = \mu_1 c + \mu_2 (1-c) \qquad (6)$$

where subscripts 1 and 2 denote the two fluids (liquid and gas). If one CV is partially filled with one and partially with the other fluid (i.e.,$0 \leq c \leq 1$), it is assumed that both fluids have the same velocity and pressure. The free surface does not represent a boundary and no boundary conditions need to be prescribed on it. The critical issue in this type of methods is the discretization of convective term in eq. (3) Since c must obey the bounds $0 \leq c \leq 1$, one has to ensure that the scheme does not generate overshoots or undershoots, but it should also keep the interface as sharp as possible, since the fluids should not mix. As noted above, the solution domain extends over both fluids, and all the conservation equations are solved in the whole domain. At the initial time step, the discretization of c is prescribed, defining the initial location of liquid and the shape of the free surface. Eq. (3) contains only convective fluxes and the unsteady term. The only scheme which unconditionally satisfies the boundedness criterion is the first order upwind scheme; however it can not be used due to excessive numerical diffusion, which smears the interface so badly that the two fluids mix over a wide region.

On the other hand, any of the higher order schemes tends to produce over- and undershoot in the vicinity of discontinuities. One can resort total variation capturing(TVD) and essentially non oscillating (ENO) schemes. However, the interface capturing in the free surface flows has some specialties which need be considered. One comes from the fact that the convective flux out of one CV must not transport more of one fluid than is available in the donor cell. During the computation of the cell-face values of c, the orientation and the local Courant number should be taken into account of.

The sharpness of the interface without over- and undershoots can be achieved by limiting the approximation of the cell-face value to lie in the shaded area of the so called *normalized variable diagram*[8] (NVD) shown in Fig. 1. The local normalized variable c in the vicinity of the cell-center C is defined

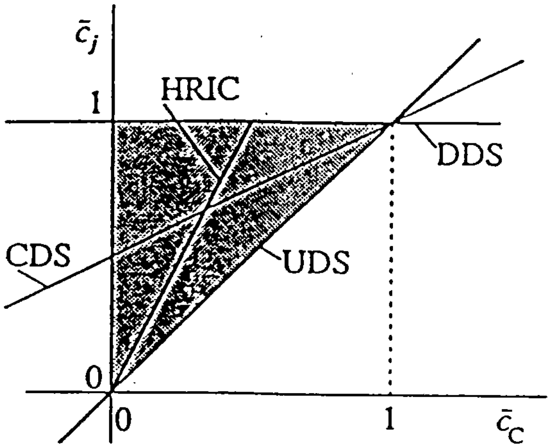

Fig. 1 Normalized variable diagram(CDS-Central Diff. Scheme, DDS-Downwind)

$$\tilde{c}(\boldsymbol{r}) = \frac{c(\boldsymbol{r}) - c_U}{c_D - c_U} \tag{7}$$

where subscripts 'U' and 'D' denote nodes upstream and downstream of the cell-center C, and $\boldsymbol{r}$ is the position vector. Should the cell-center value $\tilde{c}_C$ turn out to be smaller than zero or larger than unity, this means that the profile of c is not monotone and we need numerical diffusion to get rid of oscillations; first-order upwind scheme is then used to compute the cell-face values. For values of $\tilde{c}_C$ between zero and unity, one can choose any dependency from the shaded region of the NVD-diagram. The particular choice selected here is indicated in Fig. 1 as HRIC (high resolution interface capturing) scheme; it is a mix of a linear upwind and a downwind scheme.

The cell-face value resulting from the HRIC-scheme is further corrected according to the local Courant number and the angle between the free surface and the cell face. The aim of these corrections is to avoid stability problems at high Courant numbers (by adding more of the upwind scheme) and the occurrence of steps in the free surface. The following definitions are used.

$$C_o = \frac{\boldsymbol{v} \cdot \boldsymbol{n} S_j \Delta t}{\Delta V_c} \tag{8}$$

$$\tilde{c}_j = \begin{cases} \tilde{c}_C & \text{if} \quad \tilde{c}_C < 0.0 \\ 2\tilde{c}_C & \text{if} \quad 0.0 \le \tilde{c}_C \le 0.5 \\ 1 & \text{if} \quad 0.5 \le \tilde{c}_C \le 1.0 \\ \tilde{c}_C & \text{if} \quad 1.0 \le \tilde{c}_C \end{cases} \tag{9}$$

$$\tilde{c}^*_j = \begin{cases} \tilde{c}_j & \text{if} \quad C_o < 0.3 \\ \tilde{c}_C + (\tilde{c}_j - \tilde{c}_C)\frac{0.7 - C_o}{0.7 - 0.3} & \text{if} \quad 0.3 \le C_o \le 0.7 \\ \tilde{c}_C & \text{if} \quad 0.7 \le C_o \end{cases} \tag{10}$$

$$\tilde{c}_j^{**} = \tilde{c}_j^{*}\sqrt{\cos\theta} + \tilde{c}_C(1 - \sqrt{\cos\theta}) \tag{11}$$

Here θ represents the angle between the normal to the interface (represented by the gradient vector of c) and the normal to the cell face. Finally, the cell-face value of c is computed according to eq. (7) as;

$$c_j = \tilde{c}_j^{**}(c_D - c_U) + c_U \tag{12}$$

3. Application and Discussion

In order to demonstrate the suitability of the solution method described above, results of computations are presented. The aim of the present example is to show that the interface capturing method can treat the extreme deformations of the free surface including its

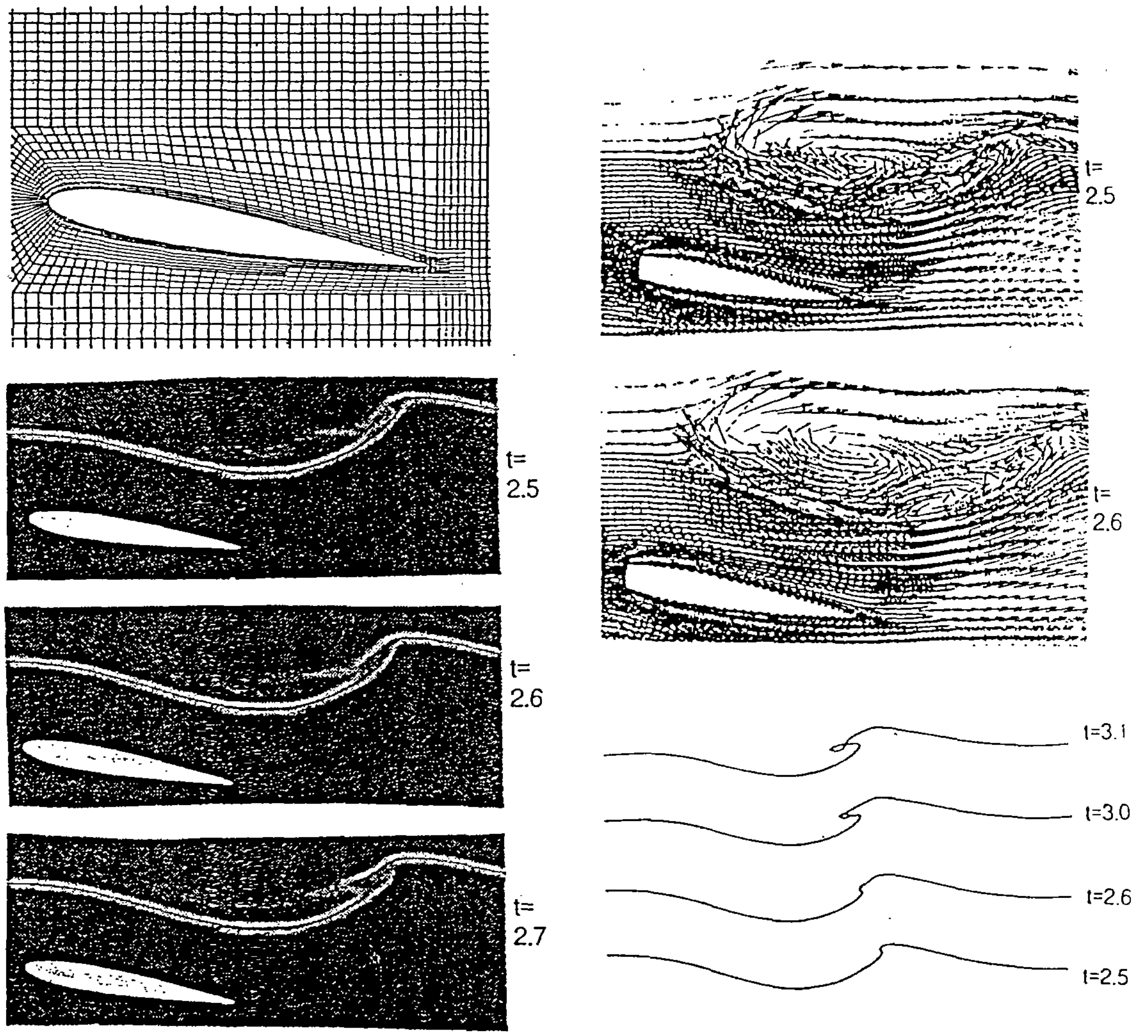

Fig. 2 Computations for laminar flows (Multigrid, Volume Fraction, Velocity, Free-Surface, from above)

overturning and air enclosures. The number of cell is 4911, maximum time step 5000, internal iteration 5, relaxation factor 0.9 and y^+ values, max. 208, min. 3.3 respectively. The computing domain is taken sufficiently in space.

The submerged hydrofoil is used for computation whose angle of attack is 10° , section shape is NACA0012, Froude number is 0.567, Reynolds number is 10^3 & 1.776×10^6, and turbulence model is of the $k-\varepsilon$ RNG type.

Fig. 2 shows the numerical results for laminar flow. The grid is of multigrid type as can be seen, which has an advantage in capturing the sharp shaped free-surface flows. It has relatively small size of grid numbers compared with MAC-based code. In the second figure we can see the free-surface shape. At t=2.7, the overturning is well simulated, which is very difficult by the FVM or FDM code. The third figures show the velocity vectors above and below the free surface. Consequently, two fluid flows can be reasonably captured by solving the interface capturing scheme. The final ones show the free-surface shape developed from t=2.5 to t=3.1. At t=3.1, the overturning waves can be numerically obrained without any difficulties.

Fig. 3 shows the numerical results for turbulent flow. In the first figure we can see the pressure distribution around the leading edge. The second figure shows the fraction of void. It is very well simulated to capture the free-surface properties. The third shows the pressure contour around the free-surface. The fourth shows the velocity vectors around the leading edge and the last shows the distribution of the kinetic energy.

From the laminar and turbulent computations, it is realized that the code can simulate any type of complex flows very reasonably. Usually the tracking method couldn't develop the overturning waves at all because it is numerically diverged at the beginning stage.

4. Concluding Remarks

The interface capturing method allows the computation of flows around bodies of arbitrary complexity, since the grid need not be adapted to the shape of the free surface. The interface capturing method presented here is applicable to flows with an arbitrary deformation of the free surface. It can capture, when the grid is locally refined, breaking waves. The results are comparable to those from the interface tracking methods with moving grids. Besides the solution method allows the use of arbitrary control

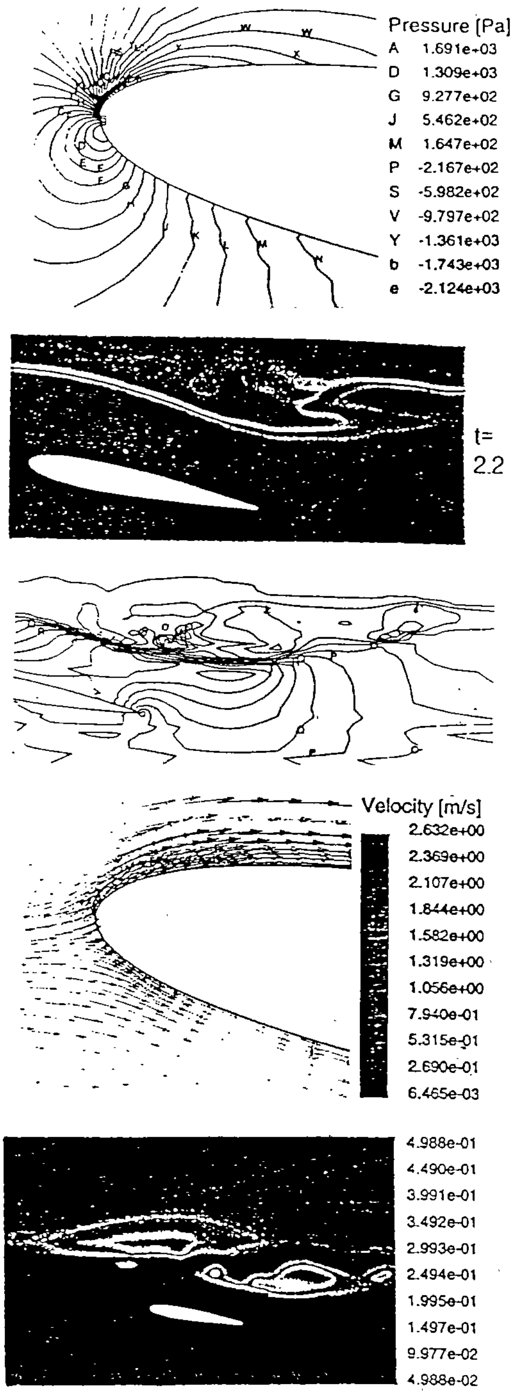

Fig. 3 Computations for turbulent flows (Pressure, Volume Fraction, Pressure, Velocity, Kinetic Energy, from above)

volumes and cell wise local mesh refinement, which is very useful for the complex flows.

요 약

물과 공기의 2층류에 대한 수치해석을 수행하였다. 임의로 변형하는 자유표면류에 2차원 수중날개를 적용하였다. 비구조격자 및 유한체적법으로 수치계산을 수행하고, 자유표면의 형상을 얻기 위하여 접면포착법을 적용하였다. 검사체적에 부분적인 격자세분화를 하였다. 공간적분은 mid point 원리, 선형내삽법, 음해법을, 시간적분은 3단계 quadratic 내삽법을 적용하였다. 선형방정식은 컬레구배법, 비선형방정식은 Picard 반복법을 사용하였며, 압력보정을 가한 운동방정식, 연속방정식으로 난류모델을 구성하였다.

References

1) Farmer, J., Martinelli, L. and Jameson, A., Fast multigrid method for solving incompressible hydrodynamic problems with free surfaces, AIAA J., Vol. 32, 1994, pp. 1175-1182.

2) Muzaferija, S. and Peric, M., Computation of free surface flows using finite volume method and moving grids, Numer. Heat Transfer, Part B, Vol. 32, 1997, pp. 369-384.

3) Harlow, F. H. and Welsh, J. E., Numerical calculation of time dependent viscous incompressible flow with free surface, Phys. Fluids, Vol. 8, 1965, pp. 2182-2189.

4) Hirt, C. W. and Nicholls, B. D., Volume of Fluid (VOF) method for dynamics of free boun- daries, J. Comput. Phys., Vol. 39, 1981, pp. 201-221.

5) Kawamura, T. and Miyata, H., Simulation of nonlinear ship flows by density-function method, J. Soc., Naval Arch., Japan, Vol. 176, 1994, pp. 1-10.

6) Ubbink, O., Numerical prediction of two fluid systems with sharp interfaces, PH. D. Thesis, Univ of London, 1997.

7) Muzaferija, S. and Peric, M., Computation of free surface flows using interface-tracking and interface-capturing methods, Chap. 3, in O. Mahrenholtz and M. Markiewicz, Nonlinear Water Wave Interaction, Computational Mechanics Publications, Southampton, 1998.

8) Leonard, B. P., Bounded higher order upwind multi-dimensional finite volume convection diffusion algolithms, in W.J. Minkowycz, E.M. Sparrow, Advances in Numerical Heat Transfer, Chap. 1, Taylor and Francis, New York, 1997, pp. 1-57.