◎ 논 문

The Efficient Algorithm for Simulating the Multiphase Flow

Seong Y. Yoon*1 and T. Yabe*2

The unified simulation for the multiphase flow by predictor-corrector scheme based on CIP method is introduced. In this algorithm, the interface between different phases is identified by a density function and tracked by solving an advection equation. Solid body motion is modeled by the translation and angular motion. The mathematical formulation and numerical results are also described. To verify the efficiency, accuracy and capability of proposed algorithm, two dimensional incompressible cavity flow, the motion of a floating ball into water and a single rising bubble by buoyancy force are numerically simulated by the present scheme. As results, it is confirmed that the present scheme gives an efficient, stable and reasonable solution in the multiphase flow problem.

Key Words: CIP Method, Time Splitting Method, Predictor-Corrector, Solid-Fluid Interaction, Level Set Method.

1. Introduction

Recently, the unified scheme for simulating the multiphase flow is taken attentions as the application field of the thermal fluid mechanics. Up to now, most of the analyses for the flow and heat transfer of multiphase fluid are mainly relied on the experimental tools. The key causes to the difficulties result from the complexity of governing equations and instability of the used scheme.

Since last decade, many numerical schemes for multiphase flow have been studied to improve the convergence and accuracy of the solutions. The studies are focused on the scheme dealing with the free surface accurately and identifying the interface. The accuracy of multiphase simulation strongly depends on capturing or tracking the interface between the different phases.

There are several representative schemes, such as a particle in cell, volume of fluid, level set method [1,2,3]. In the VOF methods [2], the interface is defined as the volume fraction of fluid. The level set method first proposed by Osher and Sethian[3] employed the distance function to identify the phase interface.

In this study, it is objective to propose the efficient algorithm for solving the multiphase flow which uses the fixed cartesian grid system, small. number of grids as possible as, and can be easily implemented into an existing algorithm.

For this purpose, the unified procedure by predictor-corrector scheme based on CIP method[4] is introduced. The CIP method first proposed by T.Yabe[5] is very accurate and less diffusive advection scheme with third order accuracy in space and then it is employed as the advection solver. The different phases are identified by the density function and the solid motion is described by the translational and rotational velocities. The Continuum Surface Force (CSF)[6] and level

* 2003년 12월 2일 접수
*1 정회원, Yeungnam University, Dept. of Mechanical Eng.
*2 비회원, Tokyo Institute of Technology

set method are employed to treat the surface tension between two different phases.

The detailed mathematical formulation and numerical results are described in next section. To verify the present scheme, two dimensional incompressible cavity flow, the motion of a floating ball into water and the single rising bubble by buoyancy force are numerically simulated by a present scheme.

2. MATHEMATICAL FORMULATION

2.1 Basic Algorithm

In this study, we have considered the time-dependent Navier-Stokes equations which can be applied on the incompressible, compressible flow[4] and also on the multiphase flow.

$$\frac{\partial \rho}{\partial t}+(\boldsymbol{u}\cdot\nabla)\rho=-\rho\nabla\cdot\boldsymbol{u} \tag{1}$$

$$\frac{\partial \boldsymbol{u}}{\partial t}+(\boldsymbol{u}\cdot\nabla)\boldsymbol{u}=-\frac{1}{\rho}\nabla p+\boldsymbol{Q}_u \tag{2}$$

$$\frac{\partial p}{\partial t}+(\boldsymbol{u}\cdot\nabla)p=-\frac{1}{\rho}\nabla\cdot\boldsymbol{u} \tag{3}$$

The governing equations can be separated into the advection and non-advection terms and the two phases are calculated by the time splitting method. The advection is simulated by the CIP method [5] with spatial accuracy of the 3rd order, while the non-advection term is discretized by a central finite difference method with the accuracy of the 2nd order in space.

Advection phase

$$\frac{\partial \rho}{\partial t}+(\boldsymbol{u}\cdot\nabla)\rho=0 \tag{4}$$

$$\frac{\partial \boldsymbol{u}}{\partial t}+(\boldsymbol{u}\cdot\nabla)\boldsymbol{u}=0 \tag{5}$$

$$\frac{\partial p}{\partial t}+(\boldsymbol{u}\cdot\nabla)p=0 \tag{6}$$

Non-advection phase

$$\frac{\rho^{n+1}-\rho^*}{\nabla t}=-\rho^*\nabla\cdot\boldsymbol{u}^{n+1} \tag{7}$$

$$\frac{\boldsymbol{u}^{n+1}-\boldsymbol{u}^*}{\Delta t}=-\frac{1}{\rho^*}\nabla p^{n+1}+\boldsymbol{Q}_u \tag{8}$$

$$\frac{p^{n+1}-p^*}{\Delta t}=-\rho^*c^2\nabla\cdot\boldsymbol{u}^{n+1} \tag{9}$$

where (*) implies the values after the advection phase.

Unfortunately, the Eqs.(7)-(9) can not be solved directly due to the existence of the values for the velocity and pressure at n+1 time level. To overcome this problem, the predictor-corrector method is introduced in this study. In the predictor step, the equation of momentum conservation is discretized in explicit formula and the predicted values are updated in the corrector step. According to the explicit formula, the momentum equation can be simply approximated using values after advection phase as follows:

$$\frac{\boldsymbol{u}^{**}-\boldsymbol{u}^*}{\Delta t}=-\frac{1}{\rho^*}\nabla p^*+\boldsymbol{Q}_u^* \tag{10}$$

where (**) represents predicted value.

By subtracting Eq.(10) from Eq.(8), we obtain the equation for velocity at n+1 time level.

$$\frac{\boldsymbol{u}^{n+1}-\boldsymbol{u}^{**}}{\Delta t}=-\frac{1}{\rho^*}\nabla\delta p \tag{11}$$

where $\delta p=p^{n+1}-p^*$

By taking a divergence of Eq.(11) and using Eq.(9) for the $\nabla\cdot\boldsymbol{u}^{n+1}$, we obtain the Poisson equation for pressure correction as

$$\nabla\cdot\left(\frac{1}{\rho^*}\nabla\delta p\right)=\frac{1}{\rho^* c_s^2\Delta t^2}\delta p+\frac{1}{\Delta t}\nabla\cdot\boldsymbol{u}^{**} \tag{12}$$

As the last stage of this algorithm, corrector step is performed using predicted values and the pressure correction, δp calculated from Eq.(12)

$$\rho^{n+1} = \rho^* - \rho^* \nabla \cdot u^{n+1} \Delta t \tag{13}$$

$$u^{n+1} = u^{**} - \frac{\Delta t}{\rho^*} \nabla \delta p \tag{14}$$

$$p^{n+1} = p^* + \delta p \tag{15}$$

If the first term on the right hand side of Eq.(12) becomes zero, this algorithm is quite similar to SMAC scheme[7]. Therefore, the present scheme may be recognized to be a generalization of SMAC to compressible fluid. In addition to the advantages, the algorithm can be applied to the multiphase flow by introducing the technique of tracking interface, the motion of solid and the surface tension.

2.2 The Technique of Tracking Interface

There are many techniques in capturing the interface such as volume of fluid, level set method and density function method. In this study, the new algorithm proposed by T.Yabe and F.Xiao[5] is used in this simulation. This algorithm introduced the density function to define the interface between two different materials and assumed the density function to be governed by the following advection equation.

$$\frac{\partial \phi}{\partial t} + (u \cdot \nabla)\phi = 0 \tag{16}$$

In order to capture the interface at discontinuous interface more effectively, we used the equation transformed by the tangent function. The transformed equation is solved by CIP method.

2.3 The Solid Motion

The behavior considered in this study can be classified into the translation and rotational motion, and the motions are modeled by the following equations.

$$\rho \frac{du}{dt} = F \tag{18}$$

$$\frac{d}{dt}(I\Omega) = \Gamma \tag{19}$$

where I, Ω and Γ are the inertial moment of a rigid body, angular velocity, and torque exerted on the solid body.

The resultant motion of solid object from the interaction of fluid is determined by

$$u_s = u + r \times \Omega \tag{20}$$

where u, r are the velocity of mass center and distance from the mass center. With the help of Eq.(20) and (16), the solid motion is calculated and the solid phase can be identified, respectively.

If the each phase is identified, the effects of different phases such as the buoyant, viscous fore and surface tension can be modelled into the last term of Eq.(2) or (8).

2.4 The Surface Tension

The surface tension on the free surface is very important in determining the shape of interface. The effect is included in the source term of the moment conservation equation. It can be written by

$$\frac{\partial u}{\partial t} + (u \cdot \nabla) u = -\frac{1}{\rho} \nabla p + \frac{1}{\rho} \sigma_s \chi \nabla \phi \tag{21}$$

where σ_s is the coefficient of surface tension. The curvature χ is calculated from

$$\chi = -\nabla \cdot n = -\nabla \cdot \frac{\nabla \phi}{|\nabla \phi|} \tag{22}$$

where ϕ is the distance function from the interface, and it is calculated by introducing the level set method proposed by Orsher and Sethian[3].

3. RESULTS OF NUMERICAL SIMULATION

In this section, the numerical results by the present scheme are shown. There are three kinds of problems that validate the present scheme and illustrate how the scheme works for the multiphase flow. First of all, the single phase flow is employed to present the possibility of the proposed scheme which calculates single and multiphase flow simultaneously and then multiphase flow problems are treated.

The accuracy of single phase flow problem is compared with other results quantitatively. In the contrary to the single phase flow, the results of the multiphase flow are focused on the performance of tracking the phase interface and conservation of property qualitatively. All the test problems are calculated on the fixed cartesian grid system and the number of grid is less than 50x50 except one case.

3.1 Two Dimensional Incompressible Flow in Driven Cavity

The calculations are performed on the uniform grid structure of 30 x 30 mesh for the Reynolds numbers of 400 and 10000 where the Reynolds number is based on the length of the square cavity and velocity of the moving wall.

The velocity profiles in the x and y direction for Re = 400 and 10000 are shown in Fig.1 where solid line and circles represent the solution by the present scheme and Ghia et al.[8], respectively.

Being compared with the results of Ghia et al., the velocity profile has shown a good agreement with those of Ghia et al. that used the multi-grid of 256 x 256.

3.2 The Motion of a Floating Ball into Water

This problem is simulating the motion of a circular ball floating into water. In this simulation, the three different kinds of phases e.g. air, liquid and solid, are considered. The density ratio of three phases is 0.01:0.5:1.0. At three walls the no slip boundary conditions are applied and free boundary at top wall. The normalized radius of ball by length of one side wall is 0.12 and the initial position is fixed at (2/3,1/3).

The floating solid ball and water affect their motion interactively. A ball is considered as rigid body and it is moving by gravitational and buoyancy forces. As shown in Fig. 2, an initially sustained ball in water moves up slowly by buoyancy force. The interaction motion of solid ball with surrounding media is modeled by the work of D.Sulsky & J.U.Brackbill[9]. To identify the each phase, the modified density function is employed. The total number of grids used in this calculation is 50 x 50.

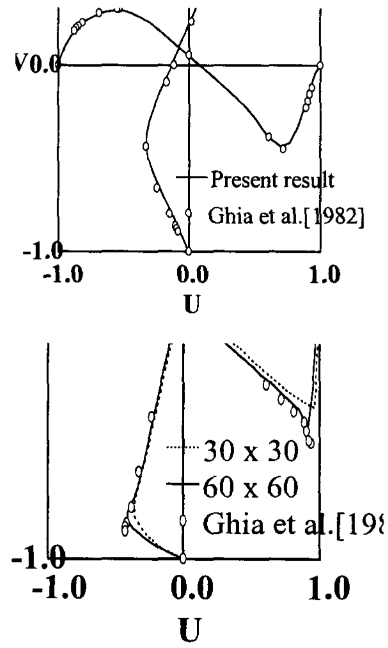

Fig.1 The comparison of the velocity profile between present results and those of Ghia et al.(256x256 multi-grids)

As shown in Fig. 2, the phase interfaces between water and air or water and solid are captured sharply. The interfaces of all the phases is identified by modified density function every time step and their motions are calculated by unified simulation procedure proposed in previous section without any internal boundary conditions.

The vertical velocity of floating ball is depicted in Fig. 3 in which positive velocity represents that the ball moves upward by buoyant force. In initial time step, the ball moves upward and downward with high amplitude, but the amplitude of the motion is decrease as the time elapses. As shown in figures 2 and 3, the oscillation of ball is well simulated by the proposed method and the shape of a circular ball is captured clearly even on the rectangular cartesian grid system. The interfaces between different phases; among the air, liquid and sold are also tracked with less numerical diffusion.

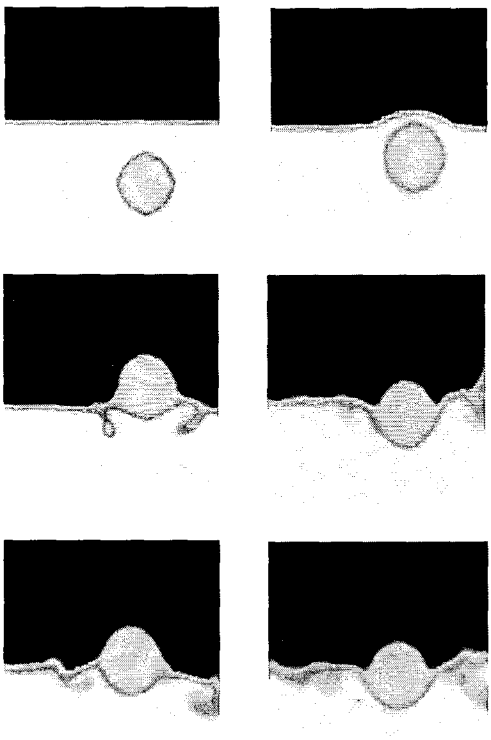

Fig. 2 Time evolution of density. Time sequence starts from left-top and ends at right-bottom(t=0.0,0.10,0.15,0.20,0.25,0.30)

3.3 The Rising Bubble by Buoyancy Force

In this calculation, the single bubble moving upward by mean of buoyancy force is simulated.

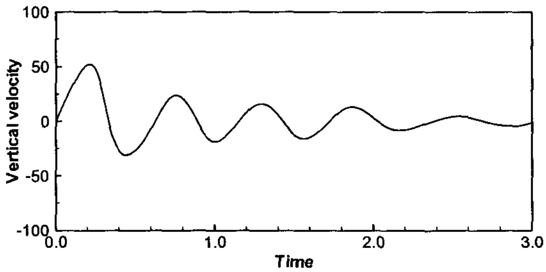

Fig.3 Time evolution of vertical velocity.

The normalized radius of ball by length of one side wall is 0.1 and the initial position is fixed at (1/2,1/4).

The bubble is changing in its shape, as it moves upward. The bubble with the normalized radius of 0.2 is initially surrounded by the water under the gravitational field and the density ratio of it to the surrounding continuum media is considered as 1:1000. Due to this high difference in the density, the conventional schemes incur the numerical diffusion or oscillation at phase interface and the stable solution cannot be obtained easily.

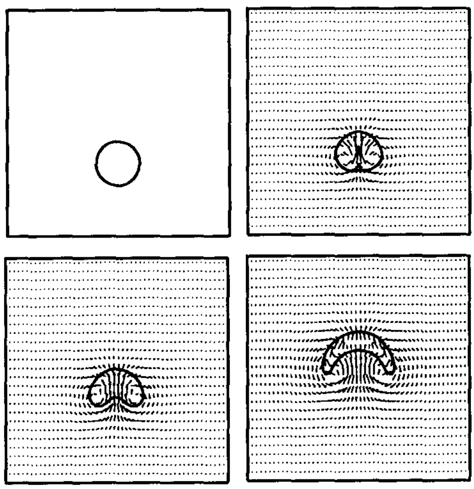

Fig.4 Time variation of free interface and velocity vector for rising bubble
a) t = 0.0 b) 0.035 c) 0.0583 d) 0.134

To overcome this problem, we used the CIP method for convection-diffusion equation and introduced the level set approach for phase interface tracking algorithm.

In Fig. 4, the evolution of phase interface and velocity vector for a single rising bubble at several time step are shown .

In this simulation, we consider the density outside the bubble as 1.0 and also density inside the bubble as 1.0x10-3. Because the difference in the density between two fluids acts as driving force, the bubble moves upward with help of the buoyancy force.

As the bubble is rising, its shape is also changing as shown in the figure. In order to calculate the shape deformation with time, the level set approach well known as front tracking method is utilized.

When the level set method for capturing the phase interface is employed, the re-initialization procedure is required to preserve the level set function.

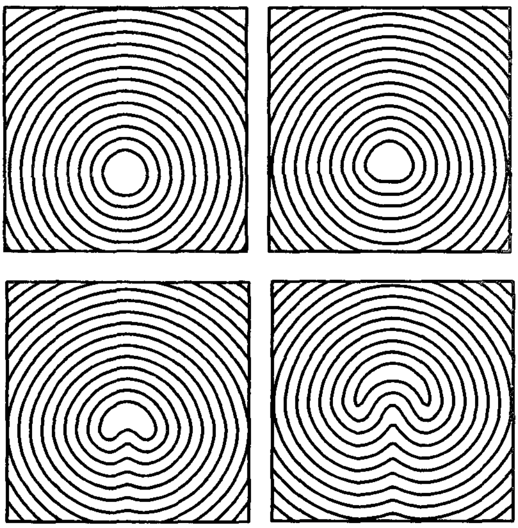

Fig.5 Profile of level set used for rising bubble simulation with re-initialization a) t = 0.0 b) 0.035 c) 0.0583 d) 0.134

In the re-initialization calculation, the modified equation is used to enhance the mass conservation. Since the conventional level set method cannot guarantee the conservation law, the zero level set function(ϕ_o) after the advection phase is modified as follows.

$$\phi_o^* = \phi_o - (A_{init} - A)/A_{init}\Delta t \quad (23)$$

where A_{init} and A are the area occupied by bubble at initial state and after advection, respectively.

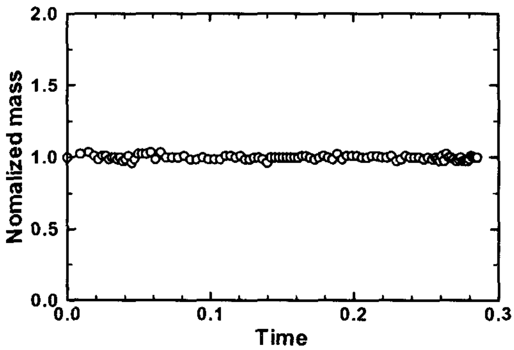

Fig.6 Trend of mass conservation for rising bubble

After the re-initialization procedure is employed to capture the interface of a single rising bubble, the profile of level set function is plotted in Fig. 5. From the figure, it can be seen that the level set approach with re-initialization treatment well describes the normal distance from phase. Fig. 6 shows the trend of mass conservation for rising bubble and the mass is non-dimensionalized by the initial mass of bubble. As shown in the figure, the mass of bubble is well conserved through the calculation.

4. CONCLUSION

In this study, the numerical scheme for multiphase flow is developed and implemented into the already developed numerical solver which is aimed to calculate the incompressible and compressible flow simultaneously[4].

To verity the proposed scheme the resultant numerical method is applied on the three problems: two dimensional cavity flow, floating ball and a single rising bubble.

In the case of a driven cavity flow, the results of present scheme have shown excellent agreements with results of Ghia et al.[8] qualitatively as well as quantitatively at all Reynolds numbers, while at high Reynolds number where the convective effects dominate the discrepancies appear near the wall. However, this departure only at the wall region is not serious considering the coarseness of the 30 x 30 meshes.

In the simulation of floating ball from water, the density function for tracking the interface of each phase is introduced. Even though the motion of a circular ball is calculated on the rectangular cartesian grid, the shape of ball is well reconstructed after long periods of complex motions From this calculation, it is found that the function identifies the three different material and captures the interface accurately.

The level set scheme combined with CIP method is applied to simulate the multi-phase flow with complex and discontinuous interface such as the rising air bubble in the continuum media under the buoyant force. The shape of bubble by a surface tension and buoyant force is simulated reasonably. The total mass of bubble is conserved sufficiently.

As a result, the present scheme can be implemented into a existing algorithm easily. Moreover it employs the one governing equation to solve the flow of multiple phases and can well capture the interface with less numerical diffusion even on a fixed rectangular cartesian grid system.

Therefore, it has been demonstrated that this scheme presented in this study is an efficient algorithm in treating the multiphase flow problem.

Acknowledgments

Author would like to appreciate the financial support provided for continuing this research by Brain Korea 21 project in 2002.

References

[1] M.W.Evans, and F.H.Harlow, "The Particle-In-Cell Method for Hydrodynamic Calculations," Los Alamos National Laboratory Report LA-2139, (1957)

[2] C.W.Hirt and B.D.Nichols, "Volume of fluid(VOF) method for the dynamics of free boundaries", Computer Physics Communications, 39, 201, (1981)

[3] S.Osher, J.A.Sethian, "Fronts propagating with curvature-dependent speed:Algorithms based on Hamilton-Jacobi formulations", Computer Physics Communications, 79, 12, (1988)

[4] Seong Y.Yoon and T.Yabe "The unified simulation for incompressible and compressible flow by the predictor-corrector scheme based on the CIP method", Computer Physics Communications, 119, (1999) pp.149-158,

[5] T.Yabe and F.Xiao, "The Constrained Interpolation Profile Method for Multiphase Analysis", Journal of Computational Physics, 169, (2001), pp.556-593.

[6] J.U.Brackbill, D.B.Kothe, and Zemach "A Continuum Method for Modeling Surface Tension", Journal of Computational Physics, 100, 335-354, (199)

[7] A.A.Amsden, and F.H.Harlow "The SMAC Metod: A Numerical Technique for Calculating Incompressible Fluid Flows", Los Alamos Scientific Laboratory Report LA-4370, (1970)

[8] U.Ghia, K.N.Ghia, and C.T.Shin, "High-Re Solution for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method", Journal of Computational Physics., 48, (1982), (1982), pp.387-411.

[9] D.Sulsky and J.U.Brackbill, "A Numerical Method for Suspension Flow", Journal of Computational Physics, 96, 339-368, (1991)