Numerical Analysis Method for Multi-dimensional Two-fluid Model using the Finite Volume Method

B.G.Huh[*a)], H.Y.Yoon [b)], D.J.Euh [b)], B.J.Yun [b)], C.-H.Song [b)], C.H.Chung [c)]
a) Korea Institute of Nuclear Safety, Yuseong, Daejeon, 305-600, Korea, [*]huha@snu.ac.kr
b) Korea Atomic Energy Research Institute, P.O.Box 105, Yuseong, Daejeon, 305-600, Korea,
c) Department of Nuclear Engineering, Seoul National University, Seoul, 157-742, Korea,

Introduction

The two-fluid model, which treats the behaviors of the gas and liquid phase separately, has been adopted in the safety evaluation codes of light water reactors (LWRs) such as the TRAC-PF1 and the RELAP5. These codes are large scale and complicated programs by considering the neutron kinetics, heat transfer in the core and the thermal-hydraulic characteristics of the components. They also need large-scale memories and long CPU times of the digital computers to calculate a two-phase flow. Because of these reasons, they can be limited in applying to the fundamental studies of the general two-phase flow dynamics.

In this study, a numerical code for the multi-dimensional two-fluid model is developed to obtain the two-phase flow parameters such as the interfacial area concentration and the void fraction. The mass, momentum equations in the two-fluid model and the number density equation are used as the governing equations. The velocity information is obtained by using the two-fluid model and the local parameters such as the void fraction and the IAC are calculated with the number density transport equation. The two-fluid model and the number density transport equation are coupled systematically with each other. In order to solve the momentum conservation equation, the constitutive relations such as the drag force, virtual mass force and lift force are considered.

The governing equations are discretized using the Finite Volume Method (FVM), which is a numerical method based on a control volume integration. An incompressible fluid is assumed and the non-staggered grid is used so that all the variables such as the pressure, velocity and temperature are defined at the cell center.

Governing equation

In order to analyze the two-phase phenomena, the multi-dimensional two-fluid model is expressed as follows:

- Mass conservation equation

$$\nabla \cdot (\phi_g + \phi_f) = 0 \tag{1}$$

- Momentum conservation equation

Gas phase

$$\rho_g \frac{D\phi_g}{Dt} = F_g - C\left|u_g - u_f\right|\left(u_g - u_f\right) - \left(\alpha_f \rho_{vm} \frac{D\phi_g}{Dt} - \alpha_g \rho_{vm} \frac{D\phi_f}{Dt}\right) \tag{2}$$

Liquid phase

$$\rho_f \frac{D\phi_f}{Dt} = F_f + C\left|u_g - u_f\right|\left(u_g - u_f\right) + \left(\alpha_f \rho_{vm} \frac{D\phi_g}{Dt} - \alpha_g \rho_{vm} \frac{D\phi_f}{Dt}\right) \tag{3}$$

where u_k, α_k and ρ_k are the velocity, void fraction and the density for each phase *k* respectively. In above equations, $\rho_{vm} = C_{vm}\left(\alpha_g \rho_g + \alpha_f \rho_f\right)$ and the term on the left-hand side indicates the time derivative which is the change of the conserved quantity ($\phi_k = \alpha_k u_k$) with time and the first term on the right-hand side includes the diffusion, pressure and body force. The second and third term of the right-hand side indicates the non-transient interfacial force and the virtual mass force respectively. The number density transport equation for each bubble group was proposed in a previous study.[1]

Finite volume method

In an analysis of two-phase flow conditions, the use of the finite difference method and the staggered grid has some problems. For example, in the boundary between the single-phase liquid and the two-phase mixture, the finite difference of the gas velocity cannot be calculated because the gas velocity does not exist in the single-phase liquid region. In order to overcome this problem, the pseudo gas with a value of the velocity is assumed even in the single-phase liquid region. Therefore, the conservation equation will not be sufficiently satisfied. In this study, the finite volume method (FVM) with a non-staggered grid is applied to solve the mass and momentum equations. In the non-staggered grid, since the velocity is not defined at the boundary surface of the control volume, the velocity, which occurs at the boundary surface with a neighboring cell, cannot be calculated automatically. Therefore, the velocity can be obtained by using the average pressure, which is given by interpolating the cell-center pressure to the pressure at the boundary surface. In order to solve the problem of a pressure oscillation, additional velocities are defined at the boundary surface of the control volume by considering a local pressure gradient. The Rhie-Chow[2] interpolation method is applied for the cell face velocity correction. The method is that the cell face velocity adds the term for considering a dissipation of the pressure numerically. The cell face velocity is given by:

$$\bar{u}_j = w\left(\bar{u}_{i1} + \frac{\Delta t V_{i1}}{\rho}\nabla p_{i1}\right) + (1-w)\left(\bar{u}_{i2} + \frac{\Delta t V_{i2}}{\rho}\nabla p_{i2}\right) - \frac{\Delta t S_j}{\rho}\left(p_{i2} - p_{i1}\right)\frac{\bar{r}_{i2i1}}{\left|\bar{r}_{i2i1}\right|} \tag{4}$$

where $\bar{u}_j$ is the cell face velocity between cell *i1* and *i2* and the third term on the right hand side suppresses

the pressure oscillation.

The governing equations are discretized using the FVM, which is a numerical method based on a control volume integration. Unstructured as well as structured grids are allowed for the application of the complex geometries. Each term appearing in the governing equations are integrated over the computing cells of Fig. 1. The first-order Euler method is used for the time integration.

$$\int_{V_i} \frac{\partial \phi}{\partial t} dv \approx V_i \frac{\phi_i^{n+1} - \phi_i^n}{\Delta t} \tag{5}$$

where ϕ_i is the conserved quantity in *i* cell.

Volume integrations of the convection, gradient, diffusion and source terms are converted into a surface integration using Gauss's theorem.

$$\int_{V_i} \nabla \cdot \bar{u}\phi dv = \int_{S_j} \bar{u}\phi \cdot \bar{n} ds \approx \sum_j \phi_j \bar{u}_j \cdot S_j \tag{6}$$

$$\int_{V_i} \nabla \phi dv = \int_{S_j} \phi \bar{i} \cdot \bar{n} ds \approx \sum_j \phi_j \bar{i} \cdot S_j \tag{7}$$

$$\int_{V_i} \nabla \cdot \nabla \phi dv = \int_{S_j} \nabla \phi \cdot \bar{n} ds \approx \sum_j \nabla \phi_j \cdot S_j \tag{8}$$

$$\int_{V_i} \phi dv \approx V_i \phi \tag{9}$$

The linear sets of the equations obtained from the proposed equations are solved using the Simplified Marker and Cell (SMAC) algorithm for the transient calculation.[3]

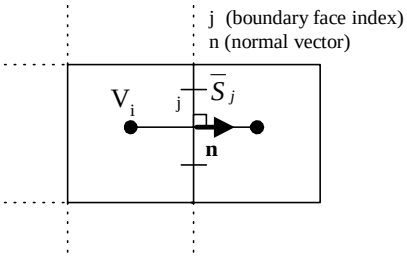

Fig. 1 The computing cell for integration

Results and discussion

To evaluate the numerical analysis method, the comparisons of the void fraction and IAC for the various flow conditions are performed against recent experimental data in air/water loop.[1] The experimental data, which is obtained at the L/D=42.2, is used as the input condition. The predictive results of void fraction and IAC at the L/D=100.7 of test section are shown in Fig. 2. As shown in the Fig. 2, the core or wall peaking for the void fraction is predicted well. The local results for the void fraction and the interfacial area concentration predict well the trends of the experimental data. Since the IAC is proportional to the void fraction, the local distribution of the IAC is similar to that of the void fraction. The predicted errors of the void fraction are 10.5%, 32.8% and 22.4%, respectively. The predicted errors of the IAC are 6.1%, 19.7% and 13.9%, respectively in three cases. In Case02, the relative deviation for the void fraction is somewhat large in the region that is near to the wall. In general, the lift, wall-lubrication and turbulent dispersion forces affect the radial distribution of the void fraction. In this study, the use of only lift force can be considered as the cause of large relative deviation. In conclusion, although the relative deviation for void fraction is large in Case02, the predictive capability of the developed code for the void fraction and IAC is acceptable in the bubbly flow regime.

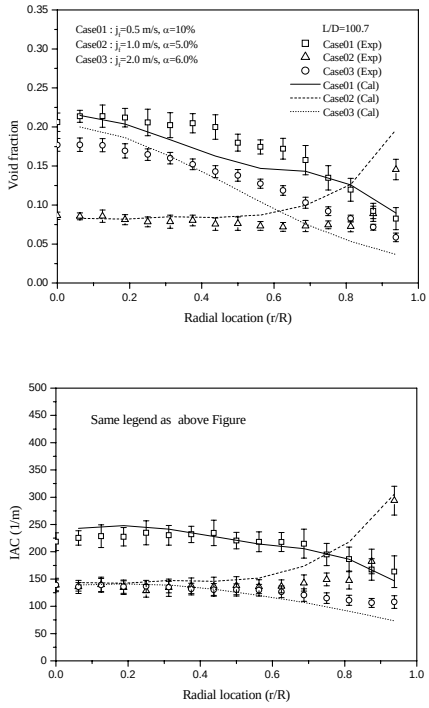

Fig. 2 Comparison results of void fraction & IAC

Conclusion

The multi-dimensional code using the finite volume method has been developed to analyze a two-phase flow. The predictive capability for void fraction and IAC seems to be reasonable and the developed code could be utilized effectively in studies of a two-phase flow.

Acknowledgement

This research has been performed under the nuclear R&D program supported by the Ministry of Science and Technology of the Korean Government.

Reference

[1] B.G Huh et al, "Prediction of Void Fraction and Interfacial Area Concentration in a Vertical Two-Phase Flow", KNS autumn meeting, 2004

[2] C.M. Rhie and W.L. Chow, "Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation", AIAA J., Vol 21, pp1525-1532, 1983

[3] J.E. Welch et al., "The MAC method - A Computing Technique for Solving Viscous, Incompressible, Transient Fluid-Flow Problems Involving Free Surface", LA-3425, 1966