Development of the Multi-dimensional Fluid Dynamics Code and Benchmarking for the Natural Circulation Problem

B.U. Bae[a*], H.Y. Yoon[b], B.G. Huh[c], D.J. Euh[b], B.J. Yun[b], C.H. Song[b], G.C. Park[a]

a Department of Nuclear Engineering, Seoul National Unviersity, Seoul,151-742, Korea,
b Korea Atomic Energy Research Institute, P.O. Box 105, Yuseong, Daejeon, 305-600, Korea
c Korea Institute of Nuclear Safety, P.O. Box 19, Yuseong, Daejeon, 305-600, Korea
* E-mail : wings21@snu.ac.kr

1. Introduction

In the analysis of two-fluid model, the interfacial area concentration (IAC) is one of the most important parameters with respect to the interfacial transfer terms between two phases. Recently, instead of the conventional static approach for modeling IAC, the IAC transport equation was developed for adiabatic bubbly flow or nucleate boiling flow[1].

This study is a part of the development of computational fluid dynamics code for analyzing the boiling flow with two-fluid model and IAC transport equation. Multi-dimensional approach was utilized in order to overcome the limitation in one-dimensional modeling for the IAC transport equation. The problem for single-phase natural circulation was used for benchmarking of the code as a first step for checking the robustness of the developed code

2. Development of the Fluid Dynamics Code

2.1 Governing equation

The governing equations for the two-phase flow analysis that are used in this study are as follows:[2]

$$\frac{\partial(\alpha_k \rho_k)}{\partial t} + \nabla \cdot (\alpha_k \rho_k v_k) = \Gamma_k \tag{1}$$

$$\frac{\partial(\alpha_k \rho_k v_k)}{\partial t} + \nabla \cdot (\alpha_k \rho_k v_k v_k) = -\nabla(\alpha_k p_k) + \nabla \cdot \left[\alpha_k \left(\overline{\overline{\tau}}_k + \tau_k^T\right)\right]$$
$$+ \alpha_k \rho_k g + v_{ki} \Gamma_k + M_{ik} - \nabla \alpha_k \cdot \tau_{ki} + p_{ki} \nabla \alpha_k \tag{2}$$

$$\frac{\partial(\alpha_k \rho_k H_k)}{\partial t} + \nabla \cdot (\alpha_k \rho_k H_k v_k) = -\nabla \cdot \left[\alpha_k \left(\overline{\overline{q}}_k + q_k^T\right)\right]$$
$$+ \alpha_k \frac{D_k p_k}{Dt} + H_{ki} \Gamma_k + a_i q_k'' + \phi_k \tag{3}$$

In discretizing the above equations, the finite volume method (FVM) was used with non-staggerd grid for the numerical stability. That is, each term in the equation was integrated over the cell volume and then defined at the center of cell.

2.2 Numerical scheme

For the advancement of the time, the explicit Euler method was used in the code. The momentum equation for each phase is solved so that the superficial velocity ($\phi_k = \alpha_k v_k$) and the pressure at next time step are calculated. For solving the matrix of pressure correction, the Simplified Marker and Cell (SMAC) algorithm was adopted with considering the term for phase change, Γ_k. And then enthalpy at the next time step is estimated in the energy equation.

2.3 Closure relations

In the momentum equation, the closure relations are necessary in terms of the interfacial momentum transfer, M_{ik}, including the drag, virtual mass force and lift force. The correlations for these are listed in the literature.[3]

For the dynamic modeling, the IAC transport equation should be implemented in the future. In this work, as a preliminary test for overall robustness of the code, the conventional correlation of Anglart et al. was utilized for IAC[4].

3. Benchmark for Natural Circulation

3.1 Definition of the problem

The problem proposed by De Vahl Davis[5] is about the natural circulation of single phase in a square as shown in Figure 1. The temperatures at left and right side are maintained as T_H and T_C, respectively, and the upper and lower side are insulated.
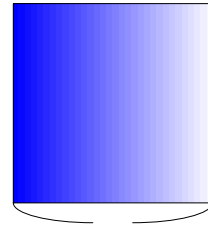

Figure 1. Geometry for benchmark problem

The test condition is given with respect to Rayleigh number defined as;

$$Ra = \frac{\rho^2 g \beta (T_H - T_C) L^3}{\mu^2} \text{Pr} \tag{4}$$

The standard solutions are known for the cases of Ra=10^3, 10^4, 10^5, and 10^6. In the application of this study, the working fluid was water at the atmospheric pressure and T_H and T_C was fixed as 80℃ and 30℃, respectively. Therefore, the size of square, L, was

determined so as to satisfy the Rayleigh number of the above value. Then the grid was generated with the structured 40X40 cells. The test geometry for four cases are summarized in Table 1.

Table 1. Test conditions

Ra	L(m)	Cell size (=L/40,m)
10^3	9.910×10^{-4}	2.478×10^{-5}
10^4	2.135×10^{-3}	5.338×10^{-5}
10^5	4.600×10^{-3}	1.150×10^{-4}
10^6	9.910×10^{-3}	2.478×10^{-4}

3.2 Results of temperature and velocity

The results of the temperature and velocity profile for the cases of Ra=10^3 and 10^6 are presented in the Figures 2, 3, 4 and 5. As shown in the results, the developed code reasonably simulates the two-dimensional behavior of natural circulation.
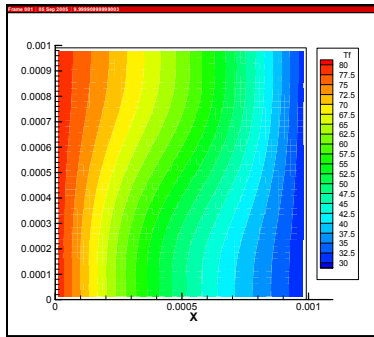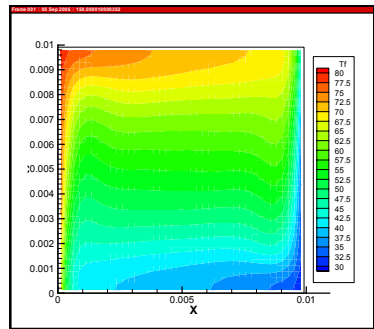

Figure 2. Temperature distribution (Ra=10^3)
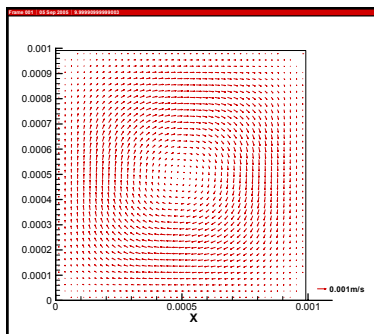

Figure 3. Temperature distribution (Ra=10^6)

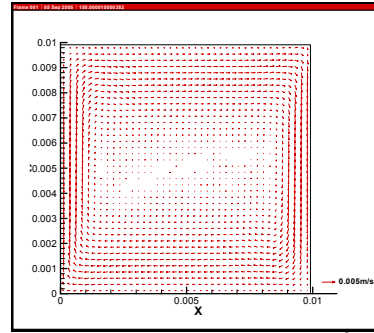Figure 4. Velocity field (Ra=10^3)

Figure 5. Velocity field (Ra=10^6)

3.3 Comparison of Nusselt number

From the temperature gradient near the wall, the average Nusselt number can be obtained as follows.

$$\overline{Nu} = \frac{1}{L} \int_0^L Nu_y dy = \frac{1}{L} \sum_i \frac{L}{T_H - T_C} \left(\frac{\partial T}{\partial x} \right)_w \Delta y \qquad (5)$$

The Nusselt numbers in this study and the literature were compared in Table 2. It shows the reasonable agreement between the developed code and the standard solution.

Table 2. Comparison of Nusselt number

Ra	CFD (Hot / Cold plate)	Reference (De Vahl Daivs, 1983)
10^3	1.130 / 1.130	1.118
10^4	2.310 / 2.310	2.243
10^5	4.903 / 4.903	4.519
10^6	10.13 / 10.12	8.8

4. Conclusion

In order to realistically analyze the two-phase flow, multi-dimensional fluid dynamics code was developed with the two-fluid model and FVM. Also, by comparing with the benchmark problem for the natural circulation, the preliminary estimation for the overall robustness of the code was evaluated.

In the future, the IAC transport equation will be implemented into the code to dynamically model the interfacial transfer terms. Then the code is expected to reasonably simulate the multi-dimensional boiling flow.

REFERENCES

[1] S. J. Kim, Research on interfacial area transport : The present and future efforts, NURETH-10, Seoul, Korea, 2003
[2] M. Ishii and et al., Two-fluid model and hydrodynamic constitutive relations, Nuclear Engineering and Design, vol. 82, pp. 107-126, 1984
[3] B. G. Huh, Experimental and analytical study of interfacial area transport in a vertical two-phase flow, Ph. D. Thesis, Seoul National University, 2005
[4] H. Anglart et al., CFD prediction of flow and phase distribution in fuel assemblies with spacers, Nuclear Engineering and Design,vol. 177, pp. 215-228, 1997
[5] G. de Vahl Davis, Natural convection of air in a square cavity: a benchmark numerical solution, Int. J. Number. Methods Fluids 3, pp. 249-264, 1983