Implementation of a Second Order Interpolation Scheme for the Convective Terms of a Two-Phase Flow Analysis Solver, CUPID

H. K. Cho[*a], H. D. Lee[b], I. K. Park[a], J. J. Jeong[a]
[a] *Korea Atomic Energy Research Institute,1045 Daedeok-daero, Yuseong-gu, Daejon, 305-353,Korea*
[b] *Korea Advanced Institute of Science and Technology,335 Gwahangno, Yuseong-gu, Daejon, 305-701,Korea*
**Corresponding author*: *hkcho@kaeri.re.kr*

1. Introduction

A two-phase flow analysis solver, CUPID, has been developed for a simulation of transient two-phase flows in light water nuclear reactor components [1]. In the CUPID solver, a two-fluid three-field model is adopted and the governing equations are solved on unstructured grids for flow analyses in complicated geometries. For the numerical solution scheme, the semi-implicit method of the RELAP5 code, which has been proved to be stable and accurate for most practical applications of nuclear thermal hydraulics, was used with some modifications for an application to unstructured non-staggered grids.

This paper is concerned with the effects of interpolation schemes for the convective terms on the simulation of two-phase flows. The calculation results with the second-order upwind scheme were compared with those with the first-order upwind difference. For the comparison, a single-phase laminar flow and a wall boiling flow were simulated. The comparison results among the two interpolation schemes apparently showed a reduced numerical diffusion with the second order scheme.

2. Second Order Upwind Interpolation Scheme for the Two-Phase Flow Solver

In our previous studies [1], the first-order upwind scheme and the second-order central difference scheme for the convective terms of the governing equations had been used for the numerical tests. So as to stabilize a numerical solution and assure a high numerical accuracy, the second-order upwind scheme was implemented into the CUPID code in the present paper.

In general, the convective terms in the current numerical solver can be expressed by

$$A\sum_f (\theta)_f (\psi_k)_f \,, \tag{1}$$

where (ψ_k): volume flow rate,
$(\theta)_f$: the convective quantities.

As indicated in Fig. 1, the convective quantities are evaluated in the first-order upwind scheme as

$$(\theta)_f = \begin{cases} \theta^- = \theta_i & if \quad (\psi)_f \geq 0 \\ \theta^+ = \theta_j & if \quad (\psi)_f < 0 \end{cases}. \tag{2}$$

In the case of the second-order upwind scheme, they are calculated by

$$(\theta)_f = \begin{cases} \theta^- = \theta_i + \Phi(\nabla\theta)_i \cdot \underline{dx}_{fi} & if \quad (\psi)_f \geq 0 \\ \theta^+ = \theta_j + \Phi(\nabla\theta)_j \cdot \underline{dx}_{fj} & if \quad (\psi)_f < 0 \end{cases}, \tag{3}$$

where $\underline{dx}_{kf} = \underline{x}_f - \underline{x}_k$,
Φ : slope limiter.

In order to obtain $\nabla\theta$ at the center of a cell, Frink's restructuring method [2] was applied which is based on the Green-Gauss method.

$$(\nabla\theta)_i = \frac{1}{V}\sum_f \overline{\theta}_f \underline{S}_f \,, \tag{4}$$

where $\overline{\theta}_f = \sum_{k=1}^{n_p} \theta_{n,k} \Big/ n_p$: a cell face value calculated by the interpolation of the face node values,
n_p : the number of nodes on the face.
The node values of a face θ_n were determined using the pseudo-Laplacian weighting method [3] as shown in Fig. 2.

In a two-phase flow, there might be a discontinuity of convective variables between two cells. To suppress the oscillation caused by the discontinuity and to assure the stability of the interpolation scheme, the slope limiter (Φ) proposed by Barth and Jesperson [4] was applied.

3. Verification of the Second Order Upwind Interpolation Scheme

To evaluate the performance of the present approach, numerical tests were performed first for both single-phase and two-phase flows: a single-phase laminar flow with a constant wall heat flux and a phase separation respectively.

Fig. 3 shows the two-dimensional computational domain and boundary conditions of the single-phase laminar flow problem. The calculations were performed with relatively coarse (5×50) and fine (10×50) meshes applying both first-order and second-order schemes so that total four calculation cases were selected for the single-phase flow example calculations..

Fig. 4 shows the temperature comparison results between the analytical solution and the calculation results of the four cases when the flow was fully developed. The comparison results showed that the present solver can capture the exact solution and more accurate results can be obtained with the second-order scheme in the same mesh.

As the second example for the verification of the second-order scheme, a wall boiling was simulated. Fig. 5 shows the two-dimensional computational domain and the initial conditions of the problem. Three meshes (6×50, 10×100 and 18×200) were used to check the mesh convergence and evaluate the performance of the second order scheme. When a steady state is achieved, the void fraction distributions at the exit of the channel were compared in Fig. 6.

As can be seen in the comparison result, a steeper shape of gas volume fraction was observed with the finer mesh and the mesh convergence of the void fraction profile was apparently established. The calculation results also showed that the second order upwind scheme yields a rather sharp void fraction profile compared to the first order scheme, which results from a reduced numerical diffusion.

From these two calculations, it was verified that the second-order upwind interpolation scheme was implemented appropriately and the numerical diffusion can be reduced with it. Moreover, it was shown that the Barth limiter, originally proposed for a single phase compressible flow, is applicable for a two-phase flow analysis.

4. Conclusion

In the present paper, the second-order upwind scheme for the convection terms was implemented for a component-scale two-phase analysis solver, CUPID. The calculations using the second-order upwind scheme indicated a reduced numerical diffusion even in a two-phase flow condition.

[1] J.J. Jeong et al., A Semi-implicit Numerical for a Transient Two-fluid Three-field Model on an unstructured grid, Int. Comm. Heat and Mass Tran., 35, pp. 597-605, 2008.
[2] N. T. Frink et al., "A Fast Upwind Solver for the Euler Equations on Three-Dimensional Unstructured Meshes," AIAA-91-0102, Reno, Nevada, 1991.
[3] N. T. Frink, "Recent Progress Toward a Three-dimensional Unstructured Navier-Stokes Flow Solver," AIAA-94-0061, Reno, Nevada, 1994.
[4] J. Barth and D. C. Jespersen, "The Design and Application of Upwind Schemes on Unstructured Meshes," AIAA-89-0366, Reno, Nevada, 1989.

Acknowledgements

This study has been carried out under the nuclear R&D program by the Korean Ministry of Education, Science and Technology.

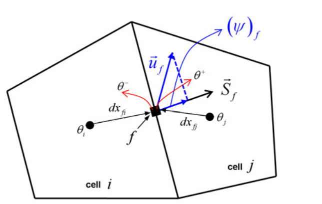

Fig. 1 Face Value Evaluation for Upwind Scheme

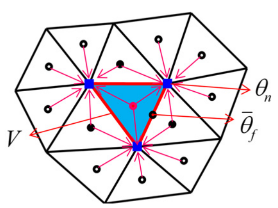

Fig. 2 Pseudo-Laplacian Weighting Method

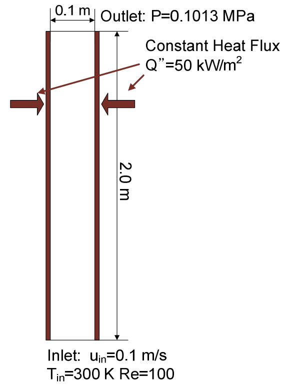

Fig. 3 Computational Domain and Boundary Conditions of the Laminar Flow Calculation

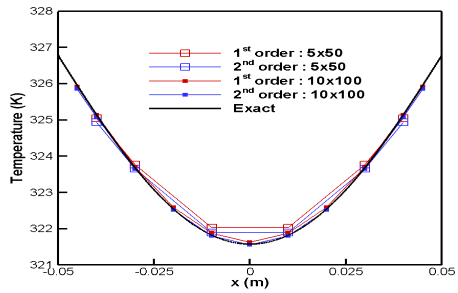

Fig. 4 Temperature Distribution Comparison Results: Exact Solution Vs Calculation Results

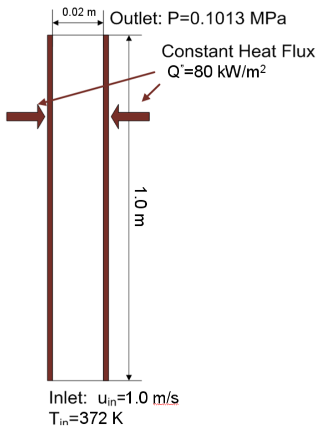

Fig. 5 Computational Domain and Boundary Conditions of the Wall Boiling Calculation

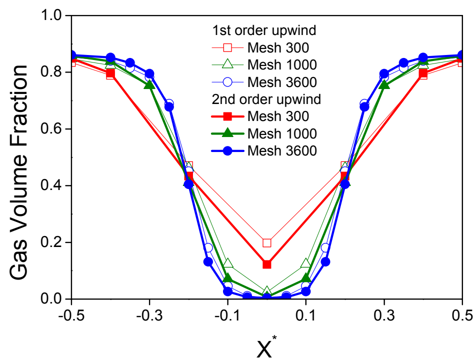

Fig. 6 Void Fraction Distribution