# Preliminary Assessment of the Component Thermal-Hydraulic Analysis Module, CUPID

H. K. Cho[*], I. K. Park, H. Y. Yoon, J. J. Jeong, J.T. Kim
*Korea Atomic Energy Research Institute,1045 Daedeok-daero, Yuseong-gu, Daejon, 305-353,Korea*
**Corresponding author: hkcho@kaeri.re.kr*

## 1. Introduction

A thermal-hydraulic analysis module, CUPID has been being developed for a design and safety analysis of a nuclear reactor component. For this purpose, a numerical method for a three-dimensional three-field model has been developed by employing the finite volume method and cell-centered scheme. The governing equations of the three-field model are solved on an unstructured mesh. As for the numerical solution scheme, the semi-implicit ICE (Implicit Continuous-fluid Eulerian) method was adopted, which has proved to be stable and accurate for most applications of a nuclear reactor transient. The conventional ICE scheme was, however, modified for an application to an unstructured mesh. The detailed mathematical models and numerical models were presented in Ref. 1.

Preliminary calculations for the unstructured semi-implicit ICE scheme have been conducted for a verification of the numerical method from a qualitative point of view. Since we focused on a qualitative test of the numerical scheme, the physical models, such as the drag coefficients, interfacial heat transfer coefficient and so on were considerably simplified and will be improved later. In the current paper, three preliminary results of the calculations are presented including 1D phase separation, 2D phase separation and 2D dam break flow.

## 2. Preliminary Assessment Results

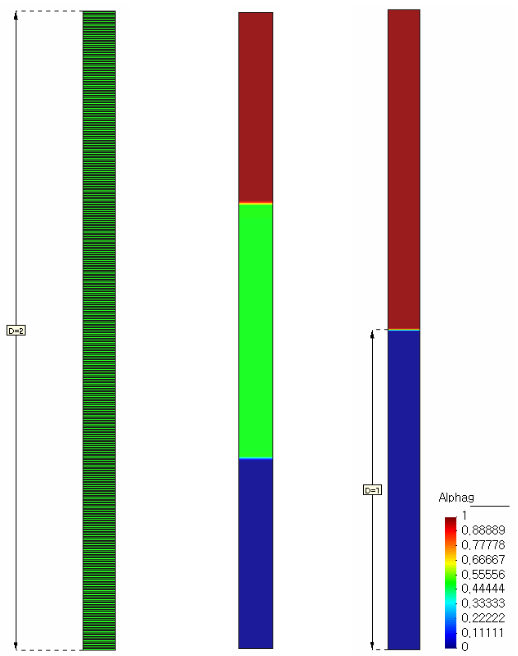
### *2.1 1-D Phase Separation*

This is an isothermal test case to investigate gravity induced phase separation. It tests the ability of the numerical scheme to predict counter-current flow conditions as exist in many reactor safety-related transients such as a flooding during a loss-of-coolant-accident. Fig.1-(a) shows the geometry and mesh of the calculation. Initially the vertical pipe is filled with a homogeneous two-phase mixture with a void fraction of $\alpha_g = 0.5$. As the calculation starts, a phase separation is induced by a density difference so that two steep void waves travels from the top and bottom ends simultaneously as shown in Fig.1-(b). The two void waves meet at the middle of a section, which results in the formation of a sharp interface after a phase separation is complete (Fig. 1-(c)).

Neglecting the momentum flux terms and virtual mass forces, we can obtain the analytical solutions of the void propagation velocities as below,

$$u_l = -\frac{\alpha_g \alpha_g (\rho_m - \rho_g) g}{F_{gl}}, \quad u_g = \frac{\alpha_g \alpha_g (\rho_m - \rho_g) g}{F_{gl}}.$$

In Fig. 2, the analytical result is compared with the calculation result. As can be seen in the comparison result, the discontinuous void fraction profile is predicted well by the present solver even though a numerically induced instability is indicated at the void wave location.




(a) t=0 s (500 meshes) (b) t=5 s (c) t=10 s

Fig.1 1-D phase separation calculation results

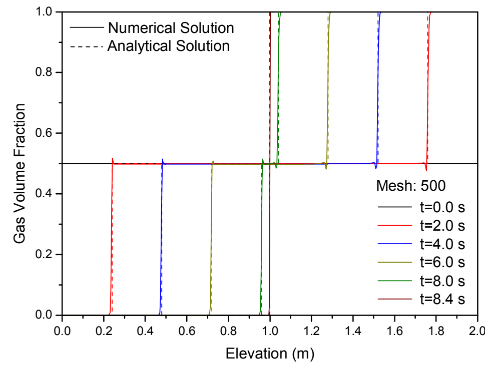
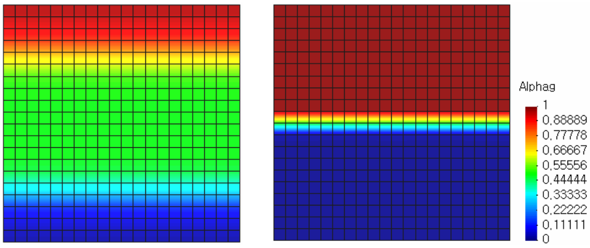



Fig.2 Analytical solution and calculation result

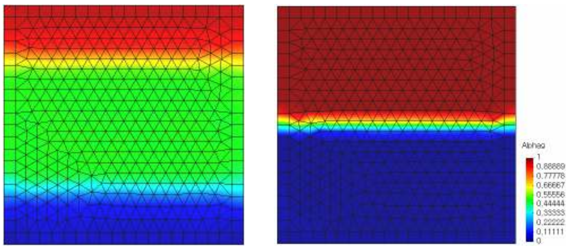
*2.2 2-D Phase Separation*

Two-dimensional phase separation calculations were conducted in structured and unstructured meshes with the same initial conditions as section 2.1. Figs. 3 and 4 show the meshes and the transients of the calculations with a structured and unstructured mesh respectively. From these calculated results, it was verified that the present solver can successfully predict the multi-dimensional sedimentation case governed by counter-current flow conditions with a unstructured mesh.




(a) t=2 s (b) t=10 s

Fig.3 2D phase separation: structured mesh

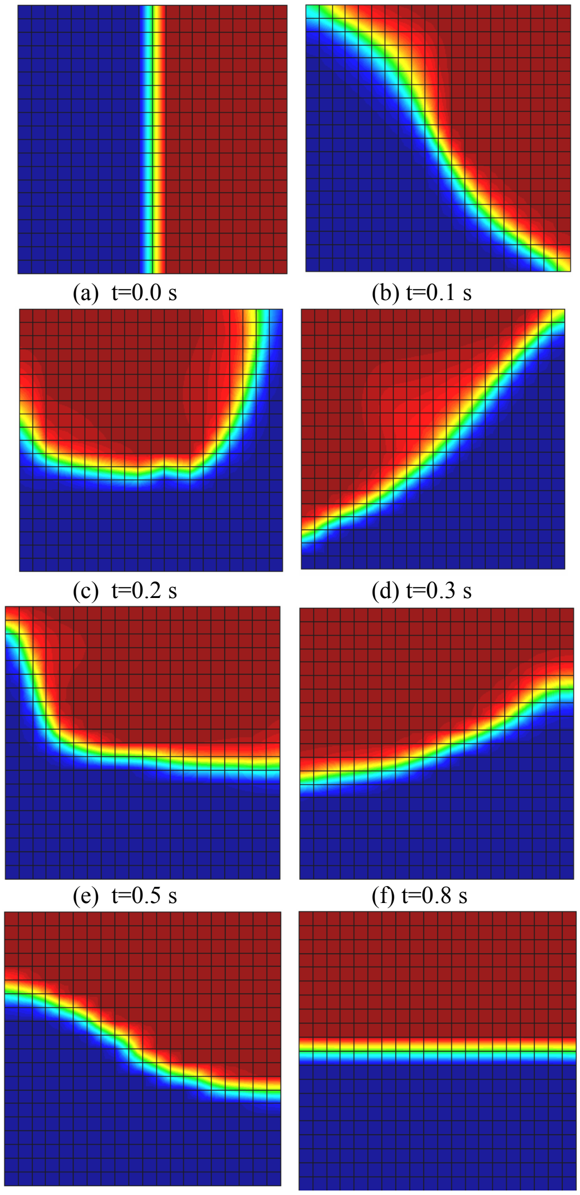


(a) t=2 s (b) t=10 s

Fig.4 2D phase separation: unstructured mesh

*2.3. 2D Dam Break Flow*

A dam break flow was simulated in order to assess the capability of the present solver for a free surface flow. The confined computation domain is a 0.1 m × 0.1 m square plane with 400 structured meshes. Initially the plane is vertically separated into liquid and gas regions by a postulated diaphragm and it is assumed to be removed instantaneously at time zero. Induced by the gravitational head, the water column collapses and eventually comes to rest. As indicated in Fig. 5, this dam break flow phenomena were qualitatively well predicted by the current solver even if it does not have an interface tracking model. It should be noted that these fluctuating interface phenomena exist in a nuclear reactor component, for example, a sweep-out in a downcomer during a LBLOCA reflood phase. For this reason, the stable prediction of the dam break flow shows the applicability of the present solver for the analysis of a nuclear reactor component.



(a) t=0.0 s (b) t=0.1 s
(c) t=0.2 s (d) t=0.3 s
(e) t=0.5 s (f) t=0.8 s
(g) t=1.0 s (h) t=4.0 s

Fig. 5 Dam break flow simulation

## 3. Conclusion

The preliminary calculation results for the verification of the CUPID solver are presented. The examples of the calculations are mostly related with a phase separation. The results showed that the present numerical scheme is robust and efficient for the modeling of phase separation phenomena.

[1] Jeong, J.J. et al., A Semi-implicit numerical for a transient two-fluid three-field model on an unstructured grid, Int. Comm. Heat and Mass Tran., to be published, 2008.