Transactions of the Korean Nuclear Society Autumn Meeting
Pyeongchang, Korea, October, 30-31, 2014

Numerical Simulation of Two-Dimensional Film Flow using CUPID1.8 Code

Y. J. Cho[a*], H. Y. Yoon[a], and H. K. Cho[b]
[a]*Korea Atomic Energy Research Institute, 111 Daedeok-daero, 989 Beon-gil, Yuseong-gu, Daejeon 305-600*
[b]*Nuclear Thermal-Hydraulic Eng. Lab. Seoul Nat. Univ. Gwanak 599, Gwanak-ro, Gwanak-gu, Seoul 151-742*
[*]*Corresponding author: yjcho@kaeri.re.kr*

1. Introduction

KAERI has developed CUPID1.8 (Component Unstructured Program for Interfacial Dynamics 1.8) code for a high-resolution analysis of two-phase flows in nuclear components [1]. Since CUPID code has been developed, various verification and validation (V&V) problems were solved to confirm not only the numerical stability, robustness and accuracy, but also the adequacy of physical models in CUPID code [2].

Recently, V&V has being performed for an interfacial drag model as well as a wall friction model. The interfacial friction plays important roles in two-phase flow problems such as a free surface in a large pool, horizontally stratified flow in cold legs, film flow in a downcomer, and so on. In particular, the interaction between the falling liquid film and the steam flow induce the two-phase multi-dimensional film flow when the emergency core coolant is injected into the upper downcomer during the reflood phase in the large break loss of coolant accident (LBLOCA). In this case, the interfacial drag force determines the reflood coolant into the core.

Two-dimensional film flow experiments performed in Korea Atomic Energy Research Institute (KAERI) was used to verify the interfacial drag model in CUPID1.8 code. In addition, the sensitivity of interfacial drag models on the film flow was also numerically investigated.

2. Two-dimensional Film Flow Experiments

2.1 Description of Experimental Facility

KAERI constructed two-phase film flow experimental facility to investigate the behavior of falling liquid film interacting with laterally blowing gas flow as shown in Fig. 1 [2]. It consists of test section made by parallel acrylic plates, an air supply and a water supply system.

Air was used instead of steam to separate the condensation effect from the hydraulic effect. The air was injected into the 6 inch pipe through a perforated plate in the expansion section for uniform flow distribution. Water with fluorescent particles for PIV was injected through a 1 inch nozzle. The injected water impinged on the test section wall (1.4 m x 0.62 m x 0.025 m) and fell down on the flat plate wall. Injected air and fallen water were divided by a separator after an interaction in the test section.

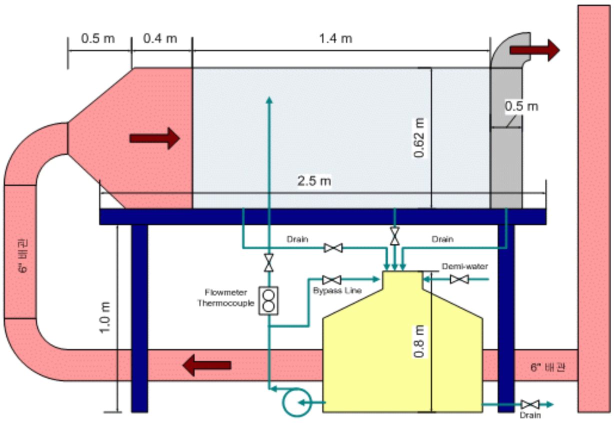

Fig. 1. Schematics of two-dimensional film flow experimental facility

2.2. Test Condition and Measurements

The test section was reduced by scaling ratio of 1/10 against APR1400. According to the linear scaling method, the velocity was scaled down by the square root of the scaling ratio. Therefore, inlet liquid velocity and air velocity were reduced to 0.63 m/s from 2 m/s and to 5 m/s ~ 15 m/s from 15 m/s ~ 45 m/s.

To measure the liquid film velocity, a volume-PIV method using the 1 to 20 μm particles was applied. The control volume was divided into 12 sub-grids to make the field of view for a high-speed camera as shown in Fig.2. Each sub-grid has dimension of 50 mm x 35 mm.

Fig. 2. Control volume for measurements

The thickness of liquid film was measured by pulse-echo type ultrasonic thickness gauge.

3. Interfacial Drag Models in CUPID1.8

CUPID1.8 uses different interfacial drag models according to the topology map. Three topology regions are used as shown in Fig. 3: bubble topology, mist topology, and sharp interface topology [3].

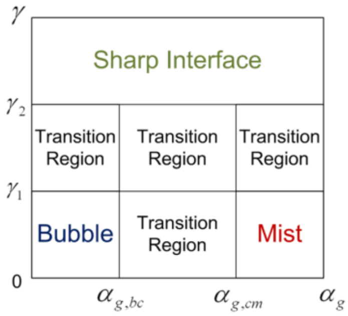

Fig. 3. Inter-phase topology map in CUPID1.8 code

If a sharp interface between a liquid and gas phase exists such as a liquid film flow, a free surface in a large pool, and a horizontally stratified flow in pipes, the drag force model for the sharp interface topology is the most sensitive factor for a stable calculation.

3.1 Bubble Topology

Interfacial drag force in bubble topology map is expressed as

$$F_{gl} = \frac{1}{8} a_i \rho_l C_D |\vec{u}_g - \vec{u}_l| \quad (1)$$

$$\begin{cases} C_D = Max\left(0.44, \dfrac{24}{\mathrm{Re}_b}\left(1+0.15\mathrm{Re}_b^{0.687}\right)\right) 0 < \mathrm{Re}_b \le 1000 \\ C_D = 0.44 \qquad \mathrm{Re}_b > 1000 \end{cases} \quad (2)$$

where, $\mathrm{Re}_b = \dfrac{\rho_l |\vec{u}_g - \vec{u}_l| D_b}{\mu_l}$.

3.2 Mist Topology

Interfacial drag force in mist topology map is expressed as

$$F_{gl} = \frac{1}{8} a_i \rho_l C_D |\vec{u}_g - \vec{u}_l| \quad (3)$$

$$\begin{cases} C_D = Max\left(0.44, \dfrac{24}{\mathrm{Re}_d}\left(1+0.15\mathrm{Re}_d^{0.687}\right)\right) 0 < \mathrm{Re}_d \le 1000 \\ C_D = 0.44 \qquad \mathrm{Re}_d > 1000 \end{cases} \quad (4)$$

where, $\mathrm{Re}_d = \dfrac{\rho_g |\vec{u}_g - \vec{u}_l| D_{drop}}{\mu_g}$.

3.3 Sharp Interface Topology

Interfacial drag force in sharp interface topology map is expressed as

$$F_{gl} = \frac{1}{2} a_i \rho_g C_i(\varphi) |\vec{u}_g - \vec{u}_l| \quad (3)$$

$$C_i(\phi) = C_{i,\tan} + (C_{i,ort} - C_{i,\tan}) |\cos\phi| \quad (4)$$

where, $C_{i,tan}$=0.005, $C_{i,ort}$=1.0, and $\cos\varphi = \dfrac{\left(\vec{u}_g - \vec{u}_l\right)\cdot\nabla\alpha}{\left|\vec{u}_g - \vec{u}_l\right| |\nabla\alpha|}$.

4. CUPID1.8 Calculation

4.1 Grid Generation

Using a commercial grid generation tool, GiD, the calculation grid was generated as shown in Fig. 4. Total 34,020 grids were used to model the water injection nozzle and the rectangular test section. Other components were treated as boundary conditions including the air blower, air injection pipe, manifold structures, and separator connected to the outlet of the test section. Finer grids were used for the water injection nozzle and the specific part of the test section where the injected water impinges on.

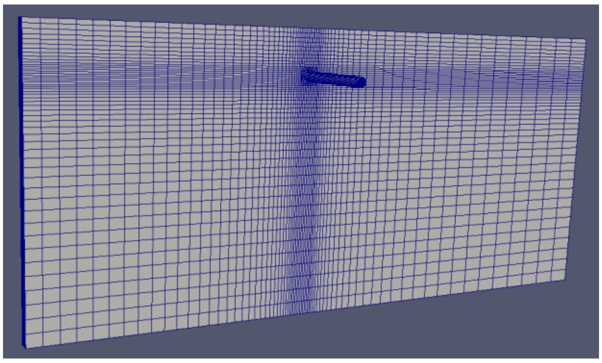

Fig. 4. Calculation grid for 2D film flow experimental facility

4.2 Initial and Boundary Conditions

As a base case, the air velocity at the inlet of test section was set to 7.0 m/s and the velocity of water injection was 0.63 m/s. Velocity profiles at the inlets were assumed to be uniform. An atmospheric pressure and room temperature air was filled in the test section and air injection nozzle at the initial state.

4.3 Calculation Results

Fig. 5 shows the void fraction at the back wall of test section. The injected water impinges on the back wall and then falls down along the wall. Liquid film moves to left side on the figure due to the air blowing from the right side.

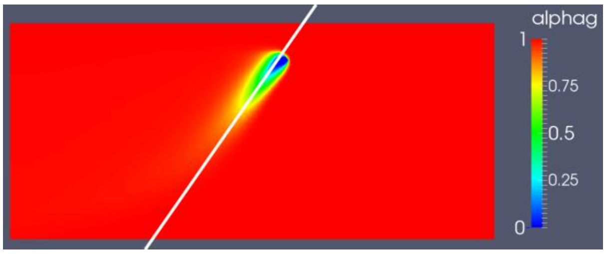

Fig. 5. Void fraction at the back wall in calculation

Fig. 6 shows the void fraction at the sliced plane along the white line in Fig. 5. This figure shows the thickness of the liquid film which has maximum value about 6 mm. while the measured liquid film thickness ranges up to 2.2 mm as shown in Fig. 7.

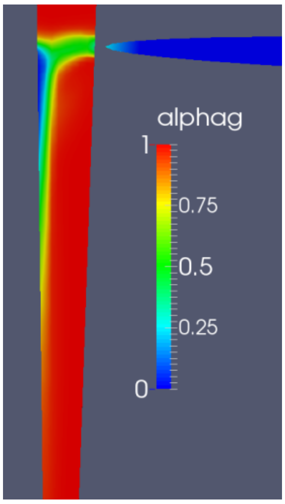

Fig. 6. Void fraction inside test section in calculation

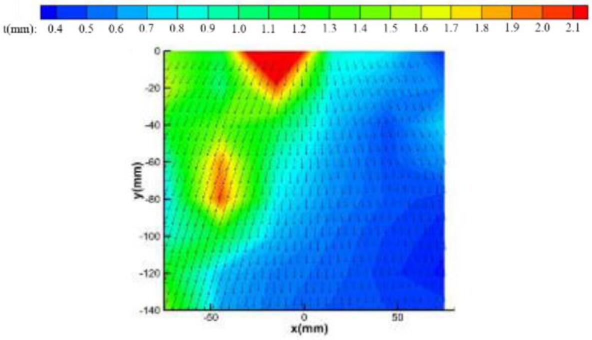

Fig. 7. Liquid film thickness in experiment

Fig. 8 and Fig. 9 show the liquid velocity vector in the calculation and experiment, respectively. The white box in Fig. 8 has identical location to the left figure in Fig. 9. As shown in two figures, calculation result significantly over-predicted the interfacial drag force so that the velocity vector has larger x-direction component compared to the experimental data.

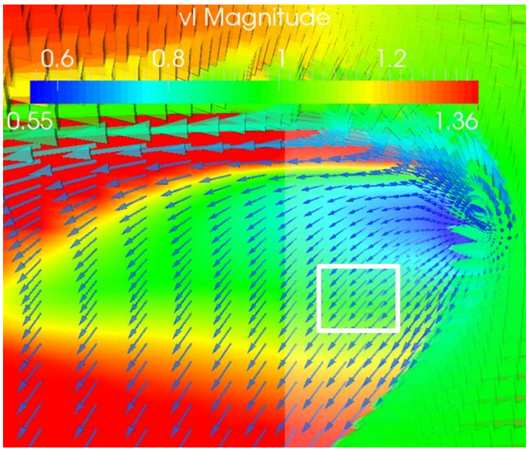

Fig. 8. Liquid velocity vector in calculation

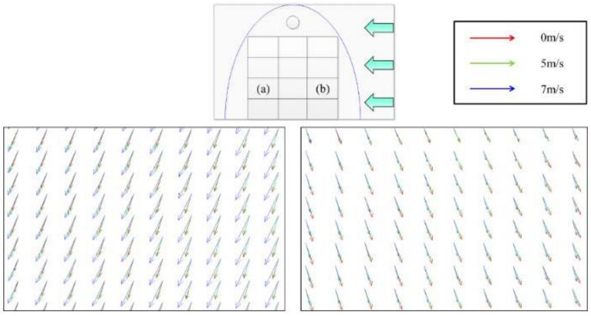

Fig. 9. Liquid velocity vector in experiment

5. Conclusions

Two-dimensional film flow experiment was simulated to assess the interfacial drag model in CUPID1.8 code. The simulation results showed that CUPID1.8 reasonably predicted the liquid film flow and its behavior due to the drag force by air flow. However, the quantities of local velocity vectors showed some discrepancies due to the large interfacial drag force in CUPID1.8 code.

This result indicates that further improvement and validation of the interfacial drag model are required. In addition, the turbulence models for each phase should be validated simultaneously since very complicated two-phase mixing phenomena occur when the injected water encounters laterally blowing air and forms steady liquid film. The turbulence model may affect velocity distributions of the liquid film and also the air.

ACKNOWLEDGMENTS

This work was supported by National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP).

REFERENCES

[1] J. J. Jeong, et al., The CUPID code development and assessment strategy, Nuclear Engineering and Technology, Vol. 42, No. 6, 2010.

[2] H. K. Cho, et al., Recent improvements to the multi-dimensional semi-implicit two-phase flow solver, CUPID, Proceedings of the 17th International Conference on Nuclear Engineering (ICONE17), July 12-16, Brussels, Belgium, 2009.

[3] J. H. Yang, H. K. Cho, G. C. Park, Experimental Study on Two-dimensional Film Flow with Lateral Air Injection, The 10th International Topical Meeting on Nuclear Thermal-Hydraulics, Operation and Safety (NUTHOS-10), Okinawa, Japan, December, 14-18, 2014.

[4] H. Y. Yoon, et al., CUPID Code Manual Volume I: Mathematical Models and Solution Methods, Korea Atomic Energy Research Institute, KAERI/TR-4403/2011.