

# Numerical Evaluation of the Cooling Performance of a Core Catcher Test Facility

Dong Hun Lee<sup>\*</sup>, Ik Kyu Park<sup>\*\*</sup>, Han Young Yoon<sup>\*\*</sup>, Kwang Soon Ha<sup>\*\*</sup>, Jae Jun Jeong<sup>\*†</sup>

<sup>\*</sup>School of Mechanical Engineering, Pusan National University,

<sup>\*\*</sup>Korea Atomic Energy Research Institute (KAERI)

(Received 3 December 2012, Revised 11 March 2013, Accepted 11 March 2013)

## Abstract

A core catcher is considered as a promising engineered system to stabilize the molten corium in the containment during a postulated severe accident in a nuclear power plant. Conceptually, the core catcher consists of a carbon steel body, sacrificial material, protection material, and engineered cooling channel. The cooling capacity of the engineered cooling channel should be guaranteed to remove the decay heat of the molten corium. The flow in ex-vessel core catcher is a combined problem of a two-phase flow in the engineered cooling channel and a single-phase natural circulation in the whole core catcher system. In this study, the analysis of the test facility for the core catcher using the CUPID code, which is a three-dimensional thermal-hydraulic code for the simulation of two-phase flows, was carried out to evaluate its cooling capacity.

**Key words** : core catcher, natural circulation, two-phase flow, CUPID code

## 1. Introduction

Nuclear power plants have various safety systems designed to prevent postulated accidents, to enhance the life time and economic benefit, and to increase public acceptance of the plants. Recently, a hypothetical severe accident, which is beyond design basis accident, is becoming a serious issue. In the current generation of nuclear power plants, there might be a chance of a severe accident. Thus, many systems have been developed against severe accidents. In particular, these systems have adopted a passive concept to operate in the case of a station black out.

A core catcher for a next-generation advanced light water reactor has been developed to stabilize the molten corium by preventing the molten corium-concrete interaction in the reactor

containment during a severe accident. As schematically shown in Fig. 1, a core catcher consists of a carbon steel body, sacrificial material, protection material, and engineered cooling channel. The molten corium discharged from a failure of the reactor vessel is collected inside the core catcher body. The sensible and decay heat of the molten corium are to be removed by the coolant channel placed on the external face of the core catcher body, where a two-phase natural circulation flow is developed [1]. The flow phenomena in the core catcher system are characterized by multi-dimensional two-phase flow behaviors with a weak driving force of the gravity. Therefore, a multi-dimensional analysis is required for investigating this natural circulation.

The multi-dimensional components of system codes and some commercial CFD codes are not applicable to practical two-phase flow problems because of their physical and numerical limitations

<sup>†</sup>To whom corresponding should be addressed.

30, Jangjeon-dong, Geumjeong-gu, Busan, 609-735  
Tel : 051-510-2455 E-mail : jjjeong@pusan.ac.kr

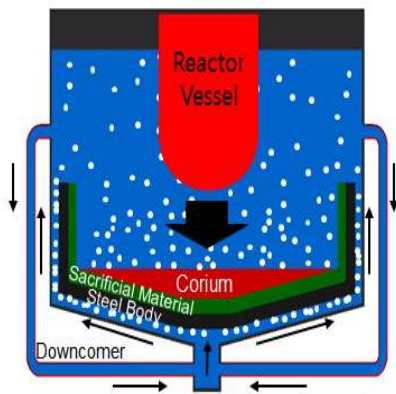


Fig. 1. Schematic diagram of the core catcher.

for simulating complicated flows with drastic phase changes [2]. In this regard, the CUPID code is a promising tool for the analysis of the core catcher system. The CUPID code has been developed by KAERI for the analysis of transient, multi-dimensional, two-phase flows in nuclear reactor components [2]. The CUPID code adopts a two-fluid, three-field model for two-phase flow. The physical modeling and numerical solution method of the CUPID code put an emphasis on versatile and robust simulations of complicated two-phase flows. This is based on a practical need to overcome the shortcomings of existing CFD codes and the multi-dimensional component of system codes.

This paper presents a preliminary analysis of the core catcher experimental facility using the CUPID code, mainly focusing on the phenomena and the heat removal capability by the natural circulation of the two-phase flow of the core catcher.

## 2. Mathematical Model of CUPID Code

### 2-1 Governing Equations

To simulate a two-phase flow, CUPID adopts a transient two-fluid, three-field model. The three fields are a continuous liquid, droplets, and a vapor. The governing equations of this model are similar to those of the time-averaged two-fluid model derived by Ishii and Hibiki [3]. The continuity,

momentum, and energy equations for the  $k$ -field are given by

$$\frac{\partial}{\partial t}(\alpha_k \rho_k) + \nabla \alpha_k \rho_k \underline{u}_k = \Gamma_k \quad (1)$$

$$\begin{aligned} & \frac{\partial}{\partial t} \alpha_k \rho_k \underline{u}_k + \nabla \cdot (\alpha_k \rho_k \underline{u}_k \underline{u}_k) \\ & = -\alpha_k \nabla P + \nabla \cdot (\alpha_k \mu_{k,eff} \nabla \underline{u}_k) + \alpha_k \rho_k \underline{g} + S_k \end{aligned} \quad (2)$$

$$\begin{aligned} & \frac{\partial(\alpha_k \rho_k e_k)}{\partial t} + \nabla \cdot (\alpha_k \rho_k e_k \underline{u}_k) \\ & = -P \frac{\partial \alpha_k}{\partial t} - P \nabla \cdot (\alpha_k \underline{u}_k) + \nabla \cdot (\alpha_k k_{c,k} \nabla T_k) + E_k, \end{aligned} \quad (3)$$

where  $\alpha_k$ ,  $\rho_k$ ,  $u_k$ ,  $P$ ,  $\Gamma_k$ , and  $e_k$  are the  $k$ -field volume fraction, density, velocity, pressure, interface mass transfer rate, and energy transfer rate, respectively.  $S_k$  represents the interfacial momentum transfer due to a mass exchange, a drag force, and non-drag forces.  $E_k$  includes the phase change, interfacial heat transfer, wall heat transfer, and volumetric heat source.

For a mathematical closure of the system of equations, constitutive relations and the equations of state are included.

To obtain numerical solutions, the finite volume method is applied to the two-fluid governing equations on unstructured grids. The solution algorithm of the CUPID code was originally developed from the semi-implicit method for a two-phase flow [4, 5] with some modifications for an application to unstructured non-staggered grids. The method was further improved for enhanced robustness and accuracy [6].

### 2-2 Physical Models and Correlations

Some of the physical models and correlations relevant to the core catcher analysis are briefly introduced in this section.

In a two-phase flow, the interfacial momentum and heat transfer strongly depend on the shape of

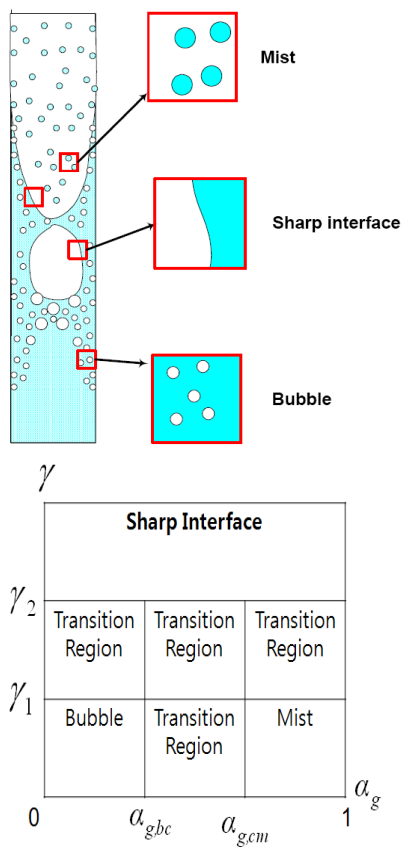


Fig. 2. Inter-phase surface topology concept.

the liquid-vapor interface. To model local interface structure of the two-phase flow, the inter-phase topology concept proposed by Tentner et al. [7] was employed in the CUPID code. They proposed the inter-phase surface topology map [8, 9]. Their works have attempted simulations of not only dispersed flows but also flows involving a local sharp interface, such as slug and annular-mist flows. In the approach, three main types of local inter-phase surface topologies and the transitional regions are distinguished, as shown in Fig. 2. The main topologies are a bubbly flow, a mist flow, and a sharp interface topology. The transitional topologies are the overlapping regions of two or three main topologies. The topology in each mesh cell is determined by two parameters, a void

fraction,  $\alpha_g$ , and a void fraction difference,  $\gamma = \delta |\nabla \alpha_g|$  [10]. Once the local topology is determined for each cell, the interfacial momentum and heat transfer models [2, 10] are calculated, depending on the topology of each cell.

In the CUPID, two models are available for the turbulent shear stress; one is the mixing length model and the other one is the standard  $k-\varepsilon$  model. The effective viscosity of the continuous liquid phase can be calculated by the sum of the bubble effect viscosity. In the case of a bubbly two-phase flow, the effect of the bubble's motion on the turbulence has been modeled as a function of the Sauter-mean bubble diameter and relative velocity, and is considered additionally [11]. The effective viscosity of the dispersed gas phase can be calculated by assuming the same kinematic viscosity of the liquid and gas.

In a subcooled boiling flow, the amount of vapor generation is computed by a wall heat flux partitioning model. The heat transfer from the wall consists of the surface quenching,  $q_q$ , evaporative heat transfer,  $q_e$ , and single-phase convection,  $q_c$ , which are explained as follows [12].

$$q = q_q + q_e + q_c \quad (4)$$

$$q_q = \left( \frac{2}{\sqrt{\pi}} \sqrt{t_w k_{c,l} \rho_l c_{p,l} f} \right) A_{2,f} (T_w - T_l) \quad (5)$$

$$q_e = N^m f \left( \frac{\pi}{6} D_d^3 \right) \rho_g h_{fg} \quad (6)$$

$$q_c = h_c A_c (T_w - T_c) \quad (7)$$

$$\Gamma_{wall} = \frac{q_e}{(h_{g,sat} - h_l)} \quad (8)$$

The wall vapor generation rate is calculated by Eq. (8). Eq. (4) is a non-linear equation with an unknown wall temperature,  $T_w$ . The equation can be

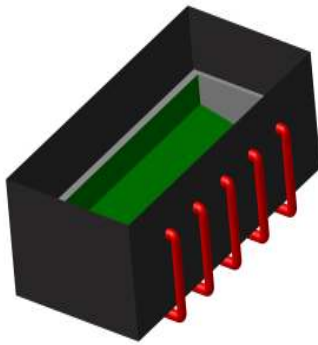


Fig. 3. Schematic diagram of the conceptual design of the prototype core catcher.

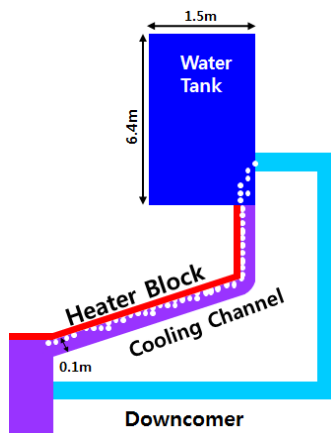


Fig. 4. Schematic diagram of the test facility.

solved by the Newton-Raphson method.

### 3. Analysis of the Core Catcher

#### 3-1 The Test Facility for the Core Catcher

The schematic diagram of the conceptual design of a prototype core catcher [13] is presented in Figs. 1 and 3. The core catcher has a rectangular shape with a dimension of 6 m x 16 m x 7 m.

To evaluate the core catcher cooling performance, KAERI has designed a test facility maintaining the thermal-hydraulic similarity with the prototype [13]. The schematic diagram of the test facility is shown in Fig. 4. It has been designed to simulate the right half of the prototype. The cooling channel of the core catcher has a gap size of 0.1 m and an angle

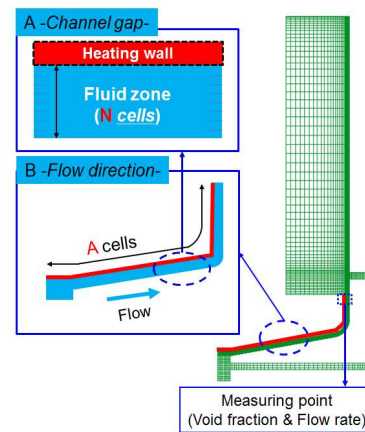


Fig. 5. Calculation domain and grid configurations for mesh sensitivity analysis : case A (channel gap), case B (flow direction).

of inclination of 10 degrees [1]. The total length of the circulation loop from the cooling channel and the downcomer is about 10 m. A water tank with a dimension 1.5 m x 6.4 m is set up on the loop.

In the subsequent analysis, it is assumed that the tank is initially filled with water at a temperature of 97 °C. The top of the tank is open to a constant pressure boundary of 0.1 MPa. The heater block is modeled as a constant heat flux boundary. The heat flux from the molten corium is assumed to be 0.133 MW/m<sup>2</sup> [13].

#### 3-2 Mesh Sensitivity Analysis

To simulate the test facility using the CUPID code, a two-dimensional grid was adopted. It is noted that a two-dimensional approach was also applied for the design of the test facility.

At first, a based input was prepared, which is based on the wide range of experiences from the CUPID code validation activities [2]. Then, the mesh sensitivity analysis on the cooling channel was carried out as shown in Fig. 5:

- (i) Case A is a sensitivity analysis on the channel gap direction. 4, 6, 8, and 10 cells were used.

(ii) Case B is on the flow direction. 85, 120, and 170 cells were used.

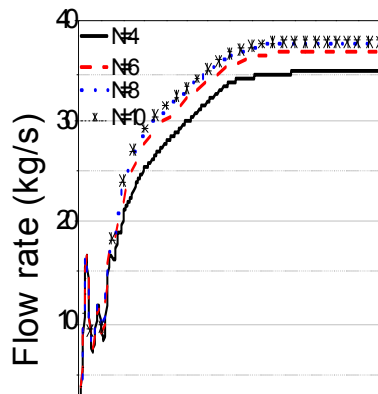


Fig. 6. Case A (channel gap) : flow rate in the cooling channel.

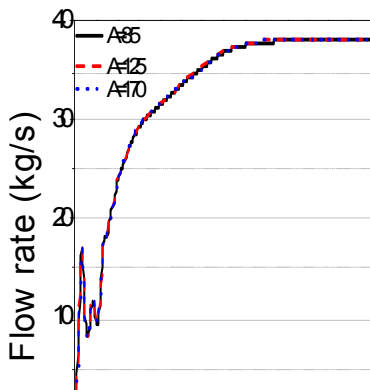


Fig. 7. Case B (flow direction) : flow rate in the cooling channel.

Figs. 6 and 7 show the calculated mass flow rate at the cooling channel, which converges to a certain value by using finer meshes. The result of case A indicates that more than 10 meshes to the channel gap direction are sufficient. Meanwhile, the result of case B shows that 85 meshes are enough to avoid mesh effects as presented in Fig. 7.

Thus, the cooling channel was discretized into a grid of 10 meshes in the gap direction and 85 meshes in the flow direction. A total computation grid with 3224 cells was used for this calculation.

It is noted that the standard k-ε model was used in this analysis.

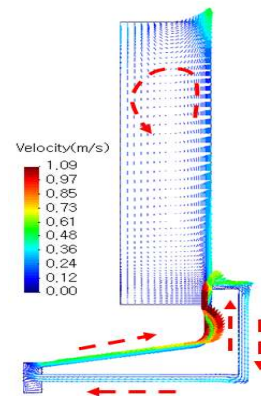


Fig. 8. Liquid velocity vector.

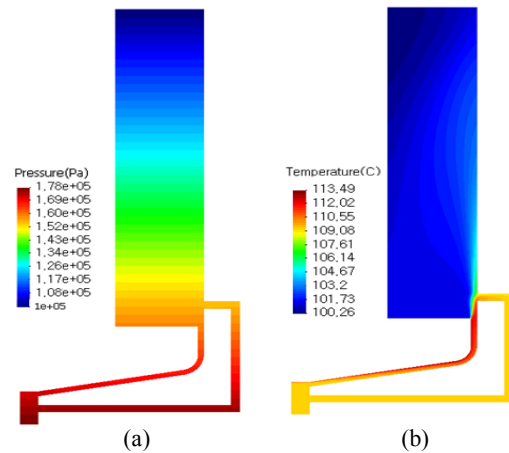


Fig. 9. Contour : (a) pressure, and (b) temperature of liquid.

### 3-3 Calculation Results

A null-transient was calculated to reach a steady state for the conditions given in section 3-1. The steady-state velocity vector is shown in Fig. 8. The flow velocity near the heated wall is faster due to a high void fraction. It seems that a two-phase natural circulation flow along the cooling channel and the downcomer is reasonably simulated considering the flow path. Fig. 8 also shows that a natural circulation flow in the water tank is formed due to the heated water flow from the cooling channel.

The contours of liquid pressure and temperature are shown in Fig. 9. The pressures are properly

calculated according to the water height as depicted

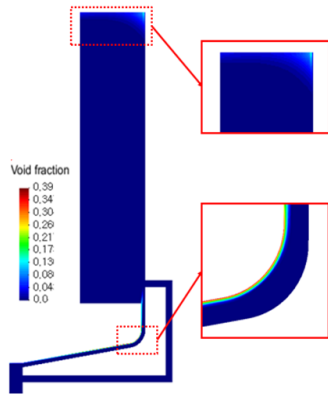


Fig. 10. Void fraction.

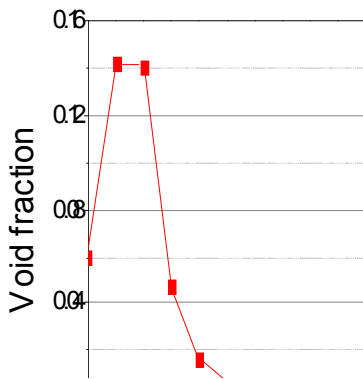


Fig. 11. Void fraction distribution of the channel outlet.

in Fig. 9 (a). The temperature near the heated wall is higher than that in any other place, as shown in Fig. 9 (b). And the liquid temperature distribution is reasonably consistent with the liquid circulation in the tank. In the upper region of the tank, the liquid temperature is slightly greater than the saturation temperature at 440 seconds as presented in Fig. 12. This resulted in a flashing, as shown in Fig. 10.

The steady-state void fraction distribution is shown in Fig. 10. Bubbles are generated at the heater block and move upwards to the water tank. However, all bubbles are condensed in the subcooled water at the right bottom of the water tank. Thus, no bubbles enter the downcomer channel.

Fig. 11 shows the void fraction distribution

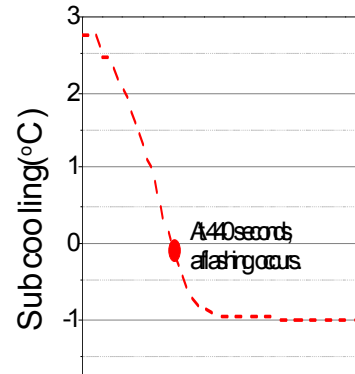


Fig. 12. Subcooling temperature of the water tank outlet.

Table 1. Comparison the analytical solution with the CUPID calculation.

	Analytical solution	CUPID calculation
Slip ratio(= $u_g/u_l$ )	2.0	~1.7
Mass flux( $\text{kg/m}^2\cdot\text{s}$ )	348.7	381.2

across the gap at the top of the heated channel. The void fraction is greater near the heated wall and gradually decreases, which is physically reasonable.

The flow rate at the downcomer is presented in Fig. 6. It shows that the circulation flow reaches a steady-state at the mass flow rate of 38.12 kg/s with a heat flux of 0.133 MW/m<sup>2</sup>. The mass flow rate in this calculation is equivalent to a mass flux of 381.2 kg/m<sup>2</sup>·s.

At present, no experimental data are available for a comparison with the calculation results. Instead, analytical solutions for the prototype were obtained by a balance between the pressure drop and hydrostatic head with some assumptions [14]. In the analysis, the mass flux was evaluated to be of 348.7 kg/m<sup>2</sup>·s with the assumption that the slip ratio at the exit of the cooling channel is 2.0. The higher slip ratio leads to a smaller void fraction, resulting in a smaller driving force and, thus, a smaller flow

rate. The two results are physically consistent and

bubbles is a dominant driving force for the flow.

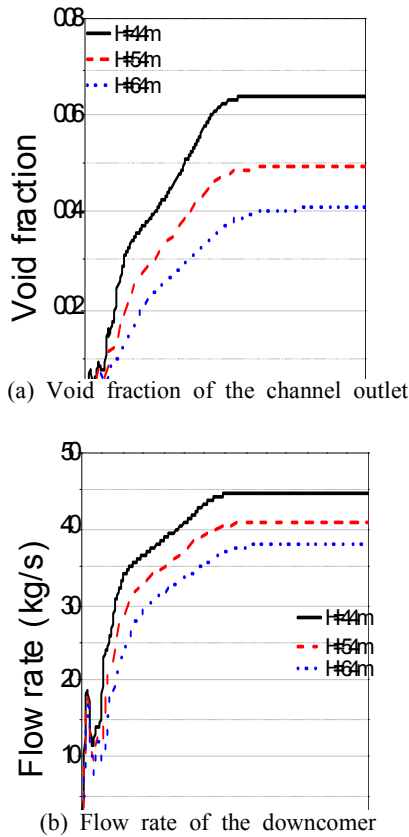


Fig. 13. Parametric study for the water tank height.

the difference between the two mass fluxes, about 10 %, is tolerable. This indicates that the core catcher analysis using CUPID code is quantitatively acceptable.

### 3-4 Parametric Studies for the Tank Height and Heat Flux

Parametric studies for the tank height and heat flux were carried out. The void fraction and the flow rate versus the height are shown in Fig. 13. As the tank height decreases, the void fraction increases because of a lower pressure in the heated channel, that is, a lower saturation temperature. This leads to a greater flow rate at the cooling channel, because the buoyancy force of vapor

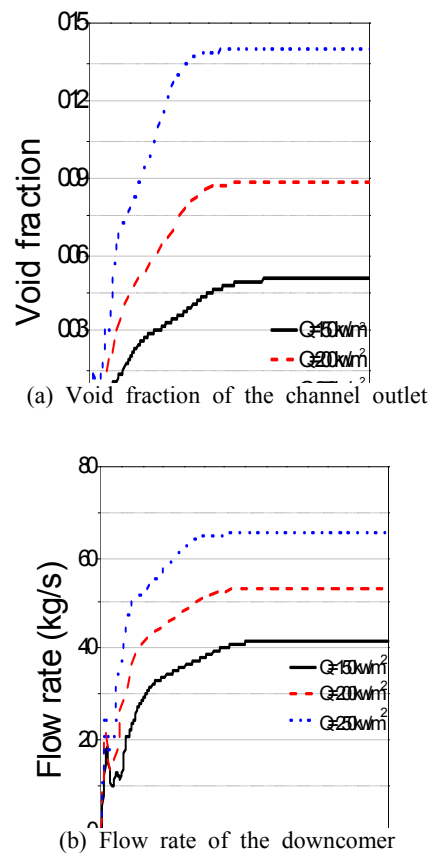


Fig. 14. Parametric study for the heat flux.

Fig. 14 shows similar results; a higher heat flux leads to a greater void fraction and mass flow rate as can be expected. The results are physically reasonable, indicating the applicability of the CUPID code for the core catcher analysis.

## 4. Conclusions

The core catcher is an engineered safety system to stabilize the molten corium in the containment during a postulated severe accident in a nuclear power plant. Its performance is characterized by two-phase natural circulation and heat transfer with drastic phase change. In this paper, the CUPID code was used for a preliminary analysis of the core

catcher performance.

The calculation results showed that the CUPID code reasonably simulates the two-phase flow phenomena in the core catcher. The calculated mass flux at the cooling channel was similar to that of an analytical solution obtained with some assumptions. The two results were physically consistent and the difference between the two mass fluxes, about 10 %, was tolerable. This indicates that the CUPID code is applicable to core catcher analysis.

The results of the CUPID calculations showed the suggested core catcher has sufficient heat removal capability by establishing the stable two-phase natural circulation. The parametric study for the tank size and wall heat flux showed that the two parameters can significantly change the characteristics of the core catcher.

#### ACKNOWLEDGEMENTS

This work was supported by the Nuclear Research and Development Program of National Research Foundation of Korea (NRF) grant funded by the Korean government (MEST) and Pusan National University Research Grant, 2011.

#### NOMENCLATURE

$A_i$	: interfacial area concentration ( $\text{m}^2/\text{m}^3$ )
$c_p$	: specific heat ( $\text{J}/\text{kg}\cdot\text{K}$ )
$e$	: internal energy ( $\text{J}/\text{kg}$ )
$E$	: interfacial energy transfer rate ( $\text{J}/\text{m}^3\cdot\text{K}$ )
$f$	: frequency (1/s)
$S$	: interfacial momentum transfer term ( $\text{N}/\text{m}^3$ )
$g$	: gravity acceleration ( $\text{m}/\text{s}^2$ )
$h$	: enthalpy( $\text{J}/\text{kg}$ )
$h_c$	: heat transfer coefficient ( $\text{J}/\text{kg}$ )
$k_c$	: conductivity ( $\text{W}/\text{m}\cdot\text{K}$ )
$P$	: pressure (Pa)

$q$	: wall heat transfer ( $\text{W}/\text{m}^2$ )
$St$	: Stanton number (-)
$t$	: time (s)
$T$	: temperature (k)
$u$	: velocity ( $\text{m}/\text{s}$ )

#### Greek Letters

$\alpha$	: volume fraction (-)
$\delta$	: characteristic length (m)
$\gamma$	: void fraction difference (-)
$\Gamma$	: interface mass transfer rate ( $\text{kg}/\text{m}^3\cdot\text{s}$ )
$\mu$	: viscosity ( $\text{kg}/\text{m}\cdot\text{s}$ )
$\rho$	: density ( $\text{kg}/\text{m}^3$ )

#### Subscripts

$b$	: bubble
$d$	: droplet
$eff$	: effective
$g$	: gas
$i$	: interface
$k$	: $g, l, \text{ or } d$
$l$	: liquid

#### REFERENCES

1. Song, J.H., Kim, H.Y., Ha, K.S., Kim, J.T., and Chenug, F.B.: "A Core Catcher Design for Advanced Light Water Reactors," 'Proceedings of ICAPP, Nice, France, May 2-5 (2011).
2. Jeong, J.J., Yoon, H.Y., Park, I.K., and Cho, H.K.: "The CUPID Code Development and Assessment Strategy," Nucl. Eng. Technol., 42, pp. 636-655 (2010).
3. Ishii, M. and Hibiki, T.: Thermo-Fluid Dynamics of Two-Phase Flow, Springer Inc., New York, pp. 217-242 (2006).
4. The RELAP5-3D Code Development Team: RELAP5-3D code manual volume I: Code structure, system models and solution methods, Idaho National Engineering and environmental Laboratory (2001).
5. Liles, D.R. and Reed, W.H.: "A semi-implicit method for two-phase fluid dynamics," J. Comput.



- Phys., 26, 390-407 (1978).
6. Yoon, H.Y. and Jeong, J.J.: "A continuity based semi-implicit scheme for transient two-phase flows," *Journal of Nuclear Science and Technology*, 47, No. 9, 779-789 (2010).
  7. Tentner, A., Lo, S., Splawski, A., Ioilev, A., Melnikov, V., Samigulin, M., Ustinenko, V., and Melnikova, S.: "Computational Fluid Dynamics Modeling of Two-Phase Flow Topologies in a Boiling Water Reactor Fuel Assembly," *Proceedings of ICONE 16, Orlando, FL, USA (2008)*.
  8. Tentner, A., Lo, S., Ioilev, A., Samigulin, M., and Ustinenko, V.: "Computational Fluid Dynamics Modeling of Two-Phase Flow in a Boiling Water Reactor Fuel Assembly," *Proceedings of the Mathematics and Computation, Supercomputing, Reactor Physics and Nuclear and Biological Applications, Avignon, France (2005)*.
  9. CD-adapco: STAR-CD, Ver. 4.0 Methodology Manual, CD-adapco, UK, Chapter 13 (2006).
  10. Cho, H.K., Yun, B.J., Yoon, H.Y., and Jeong, J.J.: "Assessment of the two-phase flow models in the CUPID code using the downcomer boiling experiment," *J. Nuclear Science and Technology*, 49, pp. 78-89 (2012).
  11. Lahey, R.T.: "The simulation of multi-dimensional multi-phase flows," *Nucl. Eng. Des.*, 235, pp.1043-1060 (2005).
  12. AEA: CFX-4 Solver Manual, UK (1997).
  13. Rhee, B.W., Ha, K.S., Park, R.J., Song, J.H., and Revankar, S.T.: "A scaling study of the natural circulation flow of the ex-vessel core catcher cooling system of EU-APR 1400 for designing a scale-down test facility for design verification," *Proceedings of ICAPP, Chicago, USA, June 24-28 (2012)*.
  14. Rhee, B.W., Ha, K.S., Park, R.J., and Song, J.H.: "A study on the effect of various design parameters on the natural circulation flow rate of the ex-vessel core catcher cooling system of EU-APR 1400," *Proceedings of ICAPP, Chicago, USA, June 24-28 (2012)*.