Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 17-18, 2012

Preliminary Analysis of the Ex-vessel Core Catcher by Using the CUPID Code

Dong Hun Lee[a*], Ik Kyu Park[b], Han Young Yoon[b], Jae Jun Jeong[a], Kwang Soon Ha[b]
[a]*School of Mechanical Engineering, Pusan National University, Busan 609-735, Korea*
[b]*Korea Atomic Energy Research Institute, 1045 Daedeok-daero, Yuseong-gu, Daejeon, Korea, 305-353*
[*]*Corresponding author: dhlee0224@pusan.ac.kr*

1. Introduction

A core catcher system has been developed for the next generation advanced light water reactors to stabilize the molten corium while avoiding molten core concrete interaction in the reactor containment during a hypothetical severe accident [1]. Analytical efforts and experimental efforts are also underway to demonstrate the performance of the proposed core catcher [2].

This paper presents the preliminary analysis results of the ex-vessel core catcher by using the CUPID code, which is a three-dimensional thermal-hydraulic code for the simulation of two-phase flows [3]. The flow in ex-vessel core catcher is a combined problem of two-phase flows in the core catcher lower flow path and single phase natural circulations in the upper pool with the heated wall boundary. The CUPID code seems to be the most promising tool for analyzing the performance of this ex-vessel core catcher, and thus, this study was started.

2. Mathematical Model

2.1 Governing Equations

Porous media model in the CUPID code is applied to this study to enhance convergence of calculation. Governing equations of porous media model are similar to the time-averaged two-fluid model derived by Ishii and Hibiki [4]. The continuity, momentum, energy and conduction equations for the k-phase are given by

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

$$\begin{aligned} &\frac{\partial}{\partial t}\alpha_k \rho_k \vec{u}_k + \nabla \cdot (\alpha_k \rho_k \vec{u}_k \vec{u}_k) \\ &= -\alpha_k \nabla P + \alpha_k \mu_k \nabla \vec{u}_k + \alpha_k \rho_k \vec{g} + S_k + \vec{M}_{wk} \end{aligned} \tag{2}$$

$$\begin{aligned} &\frac{\partial(\alpha_k \rho_k e_k)}{\partial t} + \nabla \cdot (\alpha_k \rho_k e_k \vec{u}_k) \\ &= -P\frac{\partial \alpha_k}{\partial t} - P\nabla \cdot (\alpha_k \vec{u}_k) + \alpha_k k_k \nabla T_k + E_k + q''_{fluid-solid} + q''_{fluid-porous} \end{aligned} \tag{3}$$

$$\begin{aligned} &\rho_{porous} C_{P,porous} \frac{\partial T_{porous}}{\partial t} \\ &= \nabla \cdot k_{porous} \nabla T_{porous} + q'''_{porous} + q''_{porous-solid} - q''_{fluid-porous} \end{aligned} \tag{4}$$

where, α_k, ρ_k, u_k, P, Γ_k, e_k are the k-phase volume fraction, density, velocity, pressure, an interface mass transfer rate, energy transfer rate, respectively. S_k represents the interfacial momentum transfer due to a mass exchange, a drag force, a virtual mass, and non-drag forces. E_k includes phase change, interfacial heat transfer and volumetric heat source. M_{wk} is wall friction term with form loss. When governing equations are discretized, porosity (γ) is applied to volumetric integral, permeability (ε) is applied to surface integral.

2.2 Physical Model and Correlations

To consider a turbulence effect, the k-ε turbulence model was implemented. The interfacial drag force, the interfacial heat transfer, and the interfacial area were defined by the local flow regime. A lift force, a wall lubrication force, and a turbulent dispersion force were implemented as non-drag forces.

In the subcooled boiling flow, the amount of vapor generation can be computed by a wall heat flux partitioning model. The heat transfer from the wall consist of the surface quenching q_q, evaporative heat transfer q_e, and single phase convection q_c which are basically included in the CFX-4 code as follows.

$$q = q_q + q_e + q_c \tag{5}$$

$$q_q = \left(\frac{2}{\sqrt{\pi}}\sqrt{t_w k_f \rho_f C_{pf}\, f}\right) A_{2f}\left(T_w - T_f\right) \tag{6}$$

$$q_e = N''\, f\left(\frac{\pi}{6}D_d^3\right)\rho_g h_{fg} \tag{7}$$

$$q_c = h_c A_{1f}\left(T_w - T_f\right) \tag{8}$$

Eqs. (5) is a non-linear equation with the unknown that is the temperature of the wall T_w. But T_w is the same value of the temperature of the solid in the porous media model. So, Eqs. (5) is calculated without iteration for T_w. The heat flux partitioning model adopted in the CUPID code is summarized in Table I.

Table I: All heat flux partitioning model

Parameter	Model
Active nucleate site density	$N'' = [185(T_w - T_{sat})]^{1.805}$
Bubble departure diameter	$D_d = 0.208\theta\sqrt{\sigma/(g\Delta\rho)}$
Bubble departure frequency	$f = \sqrt{4g(\rho_f - \rho_g)/(3D_d\rho_f)}$
Bubble waiting time	$t_w = 0.8/f$
Bubble influence factor	$K = 4$
Heat transfer coefficient	$h_c = St \cdot \rho_f C_{pf} u_f$

3. Analysis of the Core Catcher

The core catcher concept is presented in Fig. 1. In this study, the flow inside the inclined cooling channel was calculated (part in Fig. 1 marked by red line). The cooling channel in the core catcher is treated as two-dimensional with a gap size of 0.1m and an angle of inclination of 10 degrees [2]. The heat flux from the molten corium was assumed 0.2MW/m^2 [2].

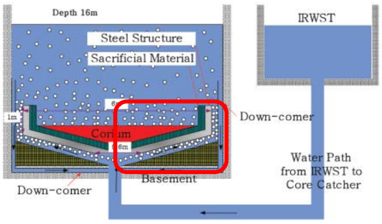


Fig. 1 Core catcher concept [2]

3.1 Analysis for the single-phase flow

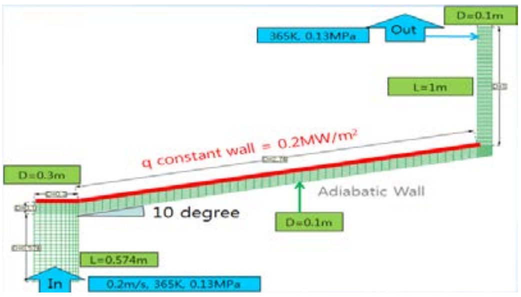
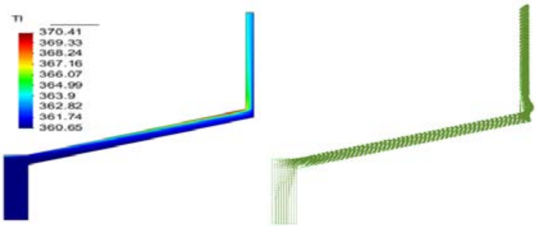


Fig. 2 Calculation domain for single-phase flow analysis

The geometrical condition and the computational mesh are presented in Fig. 2. The water of a temperature of 365K was injected to the inlet of a velocity of 0.2 m/s. The outlet set to constant pressure boundary of 0.13MPa. A computation grid with 1200 cells was used to represent the flow channel.



(a) Liquid temperature (b) Liquid velocity

Fig. 3 Single-phase flow calculation results

The contour of liquid temperature and the vector of velocity are shown Fig. 3. The temperature near the heated wall is higher value as shown in Fig. 3 (a). The velocity is properly simulated as depicted in Fig. 3 (b).

3.2 Analysis for two-phase flow

For the two-phase flow analysis, different mesh from former analysis was used to apply porous zone to heat boundary. The computational mesh is presented in Fig. 4. The water of a temperature of 375K was injected to the inlet of a velocity of 0.2 m/s. The outlet set to constant pressure boundary of 0.13MPa. A computation grid with 1800 cells was used to represent the flow channel.

The vector of velocity and the contour of volume fraction are shown Fig. 5. The flow velocity near the heated wall is faster due to interfacial drag force as shown in Fig. 5 (a). Bubbles are generated at the inclined wall and moved to outlet along the wall as depicted Fig. 5 (b).

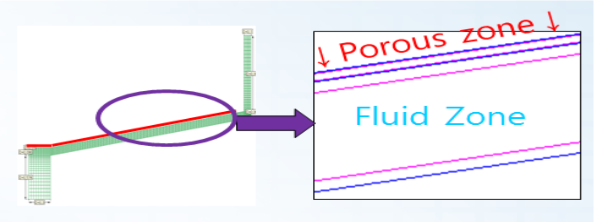
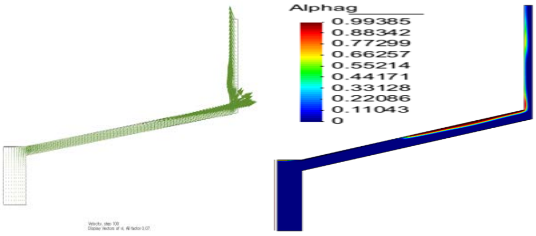


Fig. 4 Calculation domain for two-phase flow analysis



(a) Liquid velocity (b) Gas volume fraction

Fig. 5 Two-phase flow calculation results

4. Summary and Conclusions

In this study, a preliminary simulation of inclined cooling channel in the core catcher was attempted by CUPID. The simulation results showed that the CUPID code can properly perform analysis for two-phase flow. But, the further investigations about the wall heat transfer are needed for a more realistic simulation 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).

REFERENCES

[1] J. H. Song, H. Y. Kim, K. S. Ha, R. J. Park, J. T. Kim, and F. B. Cheung, A Core Catcher Design for Advanced Light Water Reactors, Proc. of ICAPP, Nice, France, May 2-5, 2011.

[2] K. S. Ha, F. B. Cheung, J. H. Song, R. J. Park, and S. B. Kim, Prediction Of Boling-Induced Natural Circulartion Flow In An Inclined Channel With Non-Uniform Flow Area, NURETH-14, Toronto, Ontario, Canada, Sep. 25-30, 2011.

[3] J. J. Jeong, H. Y. Yoon, I. K. Park, and H. K. Cho, The CUPID code Development and Assessment Strategy, Nuclear Engineering and Technology, 42(6), pp.636–655, 2010.

[4] M. Ishii and T. Hibiki, Thermo-Fluid Dynamics of Two-Phase Flow, Springer Inc., New York, p. 217, 2006.