Preliminary Simulation of PAFS Using the CUPID Code

Lee, Seung-Jun [a*], Cho, Hyoung Kyu [a], and Yoon, Han Young [a]
[a] *Thermal-Hydraulics Safety Research Div., KAERI, Daejeon, Korea*
[*] *Corresponding author: cosinesj@kaeri.re.kr*

1. Introduction

A three-dimensional thermal-hydraulics code, CUPID[1] (Component Unstructured Program for Interfacial Dynamics), is being developed by KAERI (Korea Atomic Energy Research Institute) for the component scale analyses of a reactor coolant system. The CUPID based on a two-fluid three-field model is consisted of the unstructured FVM (Finite Volume Method). The CUPID project was initiated by the requirement of a sophisticated three-dimensional thermal-hydraulics code due to recent developments of nuclear reactors. For example, the downcomer boiling of the DVI (Direct Vessel Injection) system or the acceleration of the two phase natural convection of PAFS (Passive Auxiliary Feed water System) involving extreme phase transition demands high accuracy multi-dimensional analysis techniques. In this paper, we have analyzed PASCAL model in KAERI as a preliminary simulation using the CUPID.

2. Governing Equations

The CUPID adopts a transient two-fluid three-field model in three dimensions[2]. The three-field means continuous liquid, vapor, and droplet fields. The mass, momentum and energy equations are established for the continuous liquid, vapor and droplet fields, respectively. To depict various accidents, a non-condensible gas is also considered, and it is assumed that vapor and the non-condensable gas is mixed completely. Thus, the momentum and energy equations consider vapor and non-condensible gas together while the mass equation deals with them independently. Moreover, a thermodynamic equilibrium is assumed, and so the velocity and temperature of the non-condensible gas are the same as those of the vapor.

$$\frac{\partial}{\partial t}(\alpha_k\rho_k)+\nabla\cdot(\alpha_k\rho_k\vec{\mathbf{u}}_k)=\Gamma_k, \quad (1)$$

$$\begin{aligned}\frac{\partial}{\partial t}(\alpha_k\rho_k\vec{\mathbf{u}}_k)+\nabla\cdot(\alpha_k\rho_k\vec{\mathbf{u}}_k\vec{\mathbf{u}}_k)&=\alpha_k\nabla\mathbf{P}\\ &+\nabla\cdot(\alpha_k\tau_k)+\alpha_k\rho_k\vec{\mathbf{g}}+\mathbf{P}\nabla\alpha_k+\mathbf{M}_k\end{aligned}, \quad (2)$$

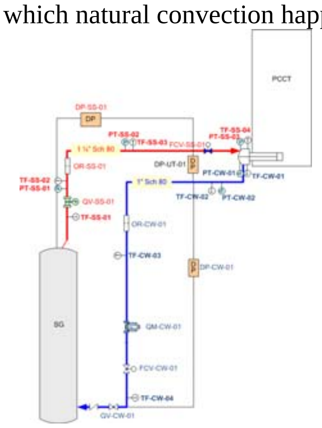
$$\begin{aligned}&\frac{\partial}{\partial t}(\alpha_k\rho_k\mathbf{e}_k)+\nabla\cdot(\alpha_k\rho_k\mathbf{e}_k\vec{\mathbf{u}}_k)=-\nabla\cdot(\alpha_k\mathbf{q}_k)\\ &+\nabla\cdot(\alpha_k\tau_k):\nabla\vec{\mathbf{u}}_k-\mathbf{P}\frac{\partial}{\partial t}\alpha_k-\mathbf{P}\nabla\cdot(\alpha_k\vec{\mathbf{u}}_k)+\mathbf{I}_k+\mathbf{Q}_k\end{aligned}, \quad (3)$$

$$\frac{\partial}{\partial t}(\alpha_v\rho_v)+\nabla\cdot(\alpha_v\rho_v\mathbf{X}_{nc}\vec{\mathbf{u}}_k)=0, \quad (4)$$

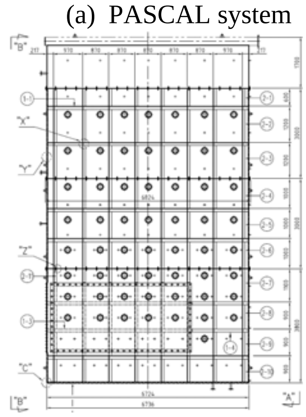
where α_k, ρ_k, $\vec{\mathbf{u}}_k$, Γ_k, $\mathbf{I}_k$ are the *k*-phase volume fraction, density, velocity, interface mass transfer rate and energy transfer rate, respectively. $\mathbf{M}_k$ represents the interfacial momentum transfer due to a mass exchange, a drag force, a virtual mass and non-drag forces.

3. PASCAL System

THSR (Thermal-Hydraulics Safety Research) division of KAERI is planning to test PAFS for the APR+ as an integral effect experiment. For the evaluation and development of thermal-hydraulic models of PAFS, PASCAL tests involving one heat exchanger tube have been done. PASCAL shows the separate effect of the PAFS for the APR+. Fig. 1 shows the PASCAL diagram. Fig. 1*a* is the conceptual diagram of the PASCAL system, which is consisted of two major systems such as a condensation heat exchanger and a PCCT. Fig. 1*b* presents the drawing of the PCCT. This paper is focusing on phenomenon in the PCCT in which natural convection happens.



(a) PASCAL system



(b) PCCT drawing

Fig.1. PASCAL system diagram.

4. Numerical Analysis

For a preliminary simulation, the CUPID assumed that the PASCAL's geometry is 6.6m×11m and the PCCT is full of water. The total heat removal is about 540 kW. Fig. 2 shows the PASCAL model discretized as 33×55. The heater is modeled as a porous media. There are differences in permeability between the horizontal pipes and the vertical pipe. Thus, although the porosity of both pipes is the same of 0.9, the permeabilities in each direction are (0.9, 1.0) and (1.0, 0.9), respectively. The volumetric heat source is uniformly distributed in porous cells. Unlike general expectations considering condensation inside the heater tube, which results in spatial transition on the amount of the heat source, the uniform heat source assumption leads the concentration of energy around the vertical tube. This makes the results different.

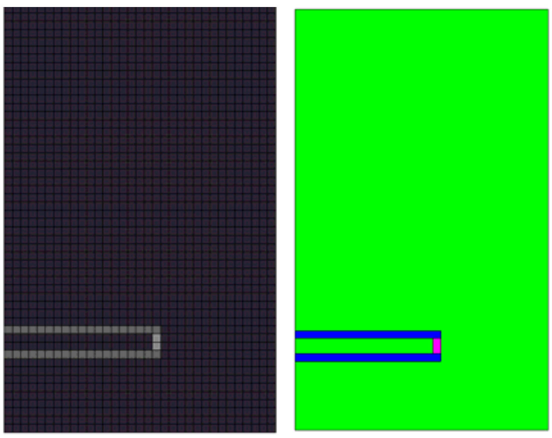


Fig. 2. PCCT model (left:grids, right:porosities)

Fig. 3 and Fig. 4 show the temperature and void fraction distribution. As time goes by, the heated tube is cooled down by natural circulation. Relatively cold water is surged into the heater area from the bottom and the heated water is risen upward and dispersed. Fig. 3 shows snap shots at t=48, 100, and 506 from the left.

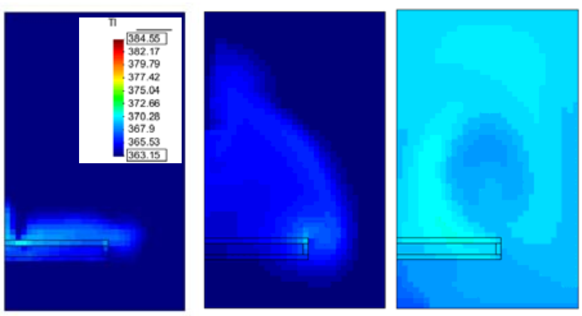


Fig. 3. Temperature distribution according to time.

Fig. 4 shows the void fraction distribution of vapor. As expected readily, the heated water evaporates continuously. The red region presents vapor phase while the blue region liquid phase.

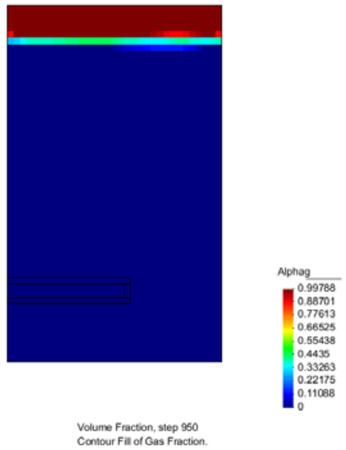


Fig.4. Gas void fraction at t=950.

5. Conclusions

In this paper, a preliminary simulation using the CUPID code has been performed about the PASCAL facility in KAERI. Even though the simulation was done in two dimensions and not considered wall drag and heat structure effects, and not included any models such as turbulence, IAT (interface area transport), lift force, and so on, natural convection phenomenon including phase transition has been presented qualitatively. To get more accurate results, appropriate models must be chosen or developed in the near future, and those works are in progress.

REFERENCES

[1] J. J. Jeong, H. Y. Yoon, I. K. Park, H. K. Cho, and H. D. Lee, Development and Preliminary Assessment of a Three-Dimensional Thermal Hydraulics Code, CUPID, NET, Vol.42, No.3, p.279, 2010.

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

.