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

Parametric study of external steam condensation in presence of air using CFD

Byong Guk Jeon [a], Hee Cheon NO[a]
[a] *Korea Advanced Institute of Science & Technology, Nuclear and Quantum Engineering Dept.,*
*Guseong-dong, Yuseong-gu, 373-1, Daejeon 305-701, Republic of Korea,*hcno@kaist.ac.kr*

1. Introduction

To ensure integrity of containment, the conventional measure against a loss of coolant accident in Advanced Power Reactor Plus (APR+) consists in spray of water drawn by pumps. These days, passive containment cooling system (PCCS) gets increased interests due to its independence to electric power supply. The performance of PCCS relies on effectiveness of heat exchangers (HXs). To predict the condensation rate of HXs, various experiments and analytical studies have been performed. In regard to analytical studies, the most common method utilizes heat and mass transfer analogy (HMTA) [1, 2]. This method is useful in fast and reliable calculation for simple geometries, mainly a single tube with vertical orientation. However, in design and arrangement of PCCS HXs, many parameters are flexible such as tube angle and diameter. Furthermore, effects of multitube degradation, containment geometry, and location of steam source or heat structures should be considered. Because HMTA cannot follow those three dimensional phenomena, CFD approach is getting more attention [3, 4]. In this study, we explore effects of tube angle, diameter, number, and chamber size after validating the CFD methodology.

2. Methods and Results

In this section, CFD calculation conditions are described. Validation of the CFD approach is briefly mentioned, and results of parametric studies are given. We used a commercial CFD code, fluent.

2.1 CFD calculation conditions

When air mass fraction is larger than 0.1, it was verified that condensate film plays negligible role in total heat transfer [5]. Therefore, condensate film is not simulated and only steam-air mixtures are simulated. Using a user defined function, condensation rate is defined on the surface of tubes in terms of mass, momentum, energy, and species sinks. From the formulation of Bird [6], we have following transport equations:

$$m''_{nc} = \rho W_{nc} v - \rho D \frac{\partial W_{nc}}{\partial n} \tag{1}$$

$$m''_{s} = \rho W_{s} v - \rho D \frac{\partial W_{s}}{\partial n} \tag{2}$$

On the interface of mixture and water film, noncondensable gas cannot penetrate into the water film. Therefore, Eq. (2) can be simplified as follows:

$$m''_{s,surface} = \frac{1}{(W_s - 1)} \rho D \frac{\partial W_s}{\partial n} \tag{3}$$

Eq. (3) represents the condensation heat flux on the condensing surface. Mass, momentum, energy, and species sink terms are defined thereafter.

$$S_{mass} = m''_{s,surface} \frac{A_{cell}}{V_{cell}} \tag{4}$$

$$S_{j-momentum} = v_j S_{mass} \tag{5}$$

$$S_{energy} = h S_{mass} \tag{6}$$

$$S_{species} = S_{mass} \tag{7}$$

Since we assume impermeable surface on the interface, a suction factor is introduced as an average of a unity and the Bird correction factor.

$$\theta = \frac{1}{2}\left(1 + \frac{\ln(1+B)}{B}\right), \text{ where } B = \frac{W_{s,i} - W_{s,\infty}}{1 - W_{s,i}} \tag{8}$$

For detailed explanation, refer to the publication by Dehbi [3].

2.2 Validation

For validation of the CFD methodology, Uchida's experimental data are benchmarked [3]. Fig.1 shows the computational domain.

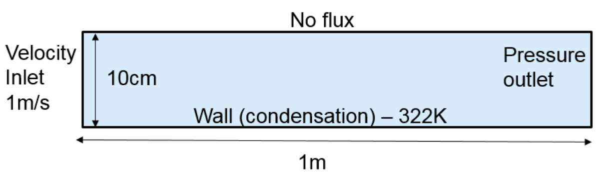

Fig. 1. Simulation domain for CFD validation.

The minimum grid size is selected as 1mm after the grid sensitivity study. For viscous models, turbulent k-ω SST model is adopted due to its better wall treatment over k-ε model. Fig.2 represents the validation results. We can see that the overall trend is well traced. It is necessary to broaden the validation cases in the future.

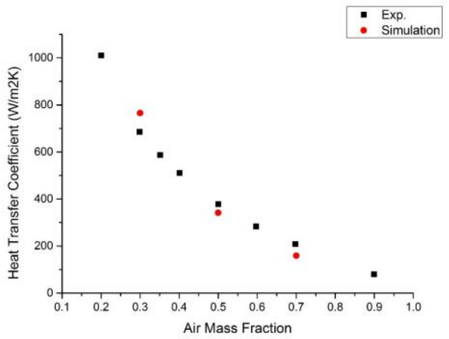

Fig. 2. Validation results of CFD calculation against experimental results from Uchida.

2.3 Parametric studies

Several parameter studies have been performed.

First, effects of various geometries for a single tube are assessed. The computational domain is shown in Fig. 3.

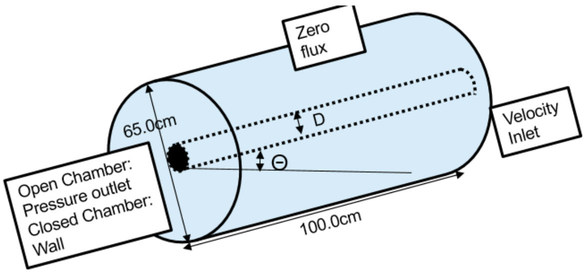

Fig. 3. Computational domain for single tube simulation.

A single tube is located in the middle of a large chamber with a diameter of 65cm. The tube length is 100cm and its surface temperature is fixed as 322K. The angle of the tube is varied from 0° to 90° and the diameter of the tube is varied from 2.45cm to 4.90cm. The inlet temperature and pressure are given as 396K, and 0.36MPa. At the opposite side of the inlet boundary, pressure outlet boundary condition is used for the open chamber and wall is disposed for the closed chamber. Steam-air mixture with 0.5 steam mass fraction is inserted at the inlet for the open chamber and pure steam is injected for the closed chamber. Fig.4 represents the effects of tube angles at various environments. We can see that regardless of chamber

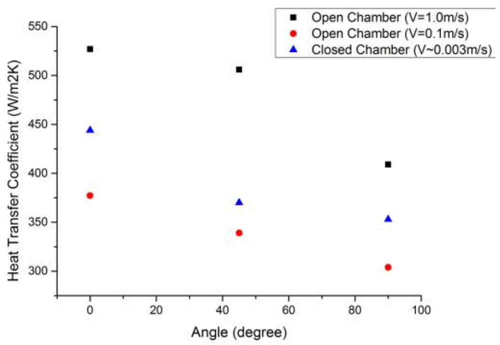

Fig. 4. Effect of tube angles on condensation.

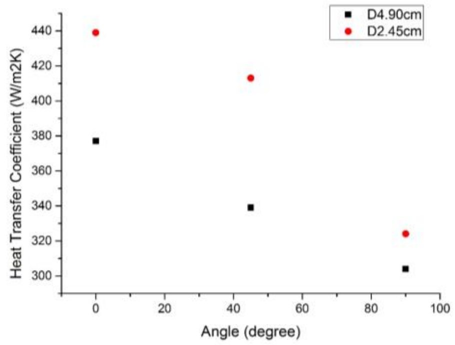

Fig. 5. Effect of tube diameter on condensation.

type and velocity, horizontal tubes give higher condensation performance. These results are in accordance with prediction by HMTA [7]. Since the simulation is conducted at fixed air mass fraction and tube wall temperature, more simulations are required and validation should be performed.

Fig. 5 displays the effect of tube diameters. We can see that the condensation is accelerated at small tube diameters.

A sample tube bundle simulation is conducted for a tube bundle comprised of nine tubes. The computational domain is given in Fig.6. The tube diameter and the distance between tubes are set up as 4.91cm and 9cm, respectively. Vertically oriented tubes are considered. The inlet mixture properties are determined same with the previous calculation.

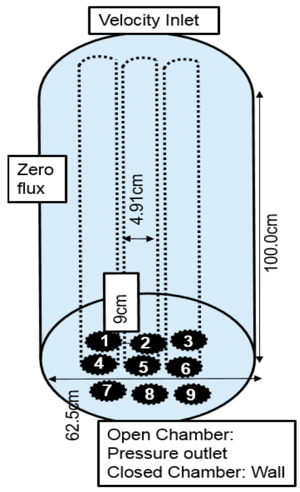

Fig. 6. Computational domain for 9 tubes simulation.

The condensation heat transfer coefficients of tubes are displayed in Fig. 7. In the figure, the tube number corresponds to the number in Fig. 6. In the open chamber, degradation is minor. On the other hand, in the closed chamber, profound degradation is observed. Since air has no way to escape in the closed chamber, air accumulates at the bottom of the chamber and steam mass fraction decreases sharply along the tube length. Comparison of steam mass fraction between the open chamber and the closed chamber is given in Fig. 8. We can see that the bulk steam mass fraction remains uniform for the open chamber while the steam mass fraction reaches even to zero at the bottom of the chamber for the closed chamber. When steam mass fraction changes a lot along the tube length, the degradation might be severe. We need to perform refined simulations for establishment of generalized models and to find out whether the actual PCCS tubes will resemble the open chamber or the closed chamber from full simulation of the containment.

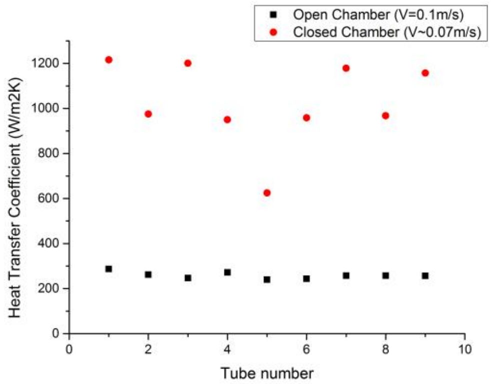

Fig. 7. Computational domain for 9 tubes simulation.

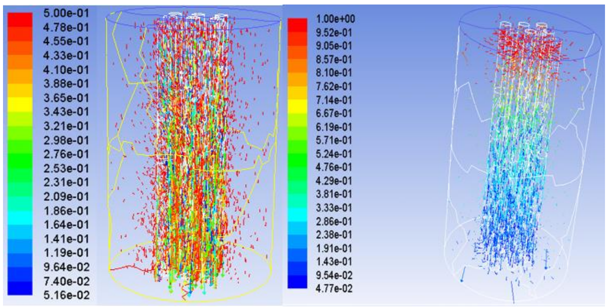

Fig. 8. Steam mass fraction of open chamber (left) and closed chamber (right).

Finally, the effect of the chamber size is evaluated. For fast calculation, 2D simulation is conducted. The models and the inlet mixture properties are retained as previous ones. The condenser is located vertically on the top of the chamber and steam source is located horizontally at the bottom. The length of condenser and steam source is maintained as 1m and 60cm. The chamber size is varied in three levels (width- height): 60cm-1m, 60cm-2m, 2.4m-8m.

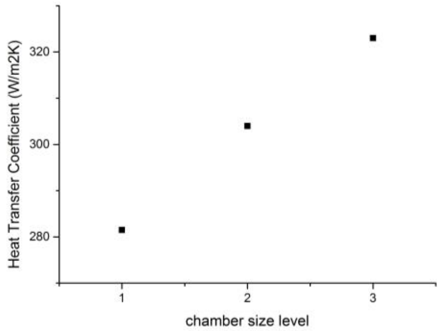

Fig. 9. Effect of chamber size on condensation heat transfer.

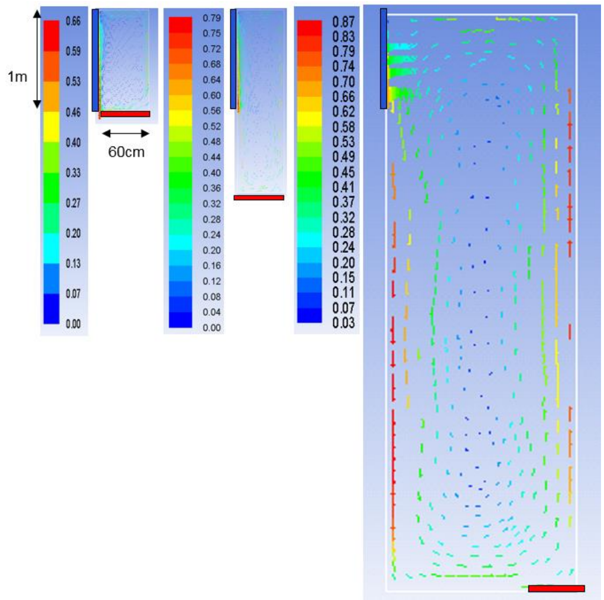

Fig. 10. Velocity profile inside chambers with different sizes.

The condensation heat transfer coefficients are given in Fig. 9. We can see that as the chamber size is increased, the heat transfer coefficient is increased.

Fig. 10 shows the velocity vector profile inside the chambers. In Fig. 10, the blue bar and the red bar represent the condenser and the steam source, respectively. We can see that the circulation trends of steam inside the chambers. The circulation rate is increased at higher chamber size due to higher height of the chamber and higher density driven force.

3. Conclusions

CFD methodology is used for consideration of 3-dimensional effects on condensation near PCCS HXs. The method is validated quite successfully by comparing with experimental results. Through parametric studies, low inclined tubes and small diameter tubes give better performance. Furthermore, multi-tube degradation becomes significant when the steam mass fraction drops a lot along the tube length. Finally, the larger chamber intensifies condensation due to higher steam-air circulation rates. Experimental validation is more required and model establishment based on systematic tests is necessary.

NOMENCLATURE

B	Bird factor (-)
D	diffusion coefficient (m^2/s) and tube diameter (m)
S	Source term (kg/m^3s for mass and species, N/m^3 for momentum, W/m^3 for energy)
W	mass fraction (-)
h	enthalpy (J/kg)
m	mass flux (kg/m^2s)
n	normal vector (m)
v	velocity (m/s)

ρ density of mixture (kg/m^3)

θ suction factor (-) and tube angle (°)

Subscripts

i interface
nc non-condensable gas
s steam
∞ bulk

ACKNOWLEDGMENTS

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea Government (MSIP) (No.2013M2A8A1038479)

REFERENCES

[1] L. E. Herranz, M. H. Andesron, and M. L. Corradini, A diffusion layer model for steam condensation within the AP600 containment, Nuclear Engineering and Design, Vol.183, pp. 133-150, 1998.
[2] A. Ganguli, A. G. Patel, N. K. Maheshwari, and A. B. Pandit, Theoretical modeling of condensation of steam outside different vertical geometries (tube, flat plates) in the presence of noncondesnable gases like air and helium, Nuclear Engineering and Design, Vol. 238, pp.2328-2340, 2008.
[3] A. Dehbi, F. Janasz, and B. Bell, Prediction of steam condensation in the presence of noncondensable gases using a CFD-based approach, Nuclear Engineering and Design, Vol. 258, pp. 199-210, 2013.
[4] L. Vyskocil, J. Schmid, and J. Macek, CFD simulation of air-steam flow with condensation, Nuclear Engineering and Design, in press, 2014.
[5] M.L. Corradini, Turbulent condensation on a cold wall in the presence of a noncondensible gas. Nuclear Technology. Vol.64, pp. 186–195, 1984.
[6] R.B. Bird, W.E. Stewart, E.N. Lightfoot, Transport Phenomena. Wiley and Sons Inc., New York, 2002.
[7] J. H. Heo and B. J. Chung, Natural convection heat transfer on the outer surface of inclined cylinders, Chemical Engineering Science, Vol. 73, pp. 366-372, 2012.