A Computational Investigation for Steam Condensation on a Flat Plate in the Presence of Non-condensable Gas Using a CUPID Code

Min-Seok Ko[a*], Yeon-Jun Choo[a], Soon-Joon Hong[a]
[a] *FNC Technology Co. Ltd, Heungdeok IT Valley, Heungdeok 1-ro, Giheung-gu, Yongin-si, Gyeonggi-do, 446-908, Korea*

1. Introduction

Steam condensation is a very important issue in various engineering fields including refrigeration, heat exchanger, and distillation system and so on. For a typical PWR which is equipped with the concrete containment, depressurization during design basis and severe accidents should be achieved by condensing steam on a cooling device called the Passive Containment Cooling System (PCCS). The PCCS consists of a number of tube banks, such that condensation occurs at the exterior surface of the tubes, and the cooling fluid comes into the tube bank from the heat sink called the Passive Containment Cooling Tank (PCCT) installed at the outside of the containment in such a way that working fluid can circulate the loop between the PCCS cooling device and PCCT in a passive way, relying only on gravitational force.

In many nuclear safety applications, steam condensation occurs in the presence of non-condensable gases. In the case of a LOCA, for example, released steam condenses on various passive heat structures in the presence of some amounts of air. For a hypothetical severe accident, hydrogen generated from core damage degrades steam condensation. For nuclear safety analyses, lumped parameter codes have been conventionally used due to less computational burden. However, since most lumped parameter codes use simplified flow models, they often over-predicts or under-predicts the thermal-hydraulic responses. In addition, flow variables are calculated over relatively large cells or lumped parameter volumes, thus, they have inherent limitations in describing flow fields. As the performance of the computer hardware and algorithms has been improved, the attempts in order to analyze the flow fields inside the large computational domain using CFD codes have been made with the aim of describing the details of 3D phenomena and hence reducing the uncertainties of the lumped parameter codes [1, 2, 3, 4].

In nuclear containment applications, the near-wall region is important because walls are treated as a sinks or sources of mass, momentum, and energy when the condensation or evaporation occurs. The wall-function approach is a practical tool to model the near-wall phenomena in large volumes. In this method, near-wall region up to $y^+ \sim 30$ is bridged using the standard wall function instead of resolving the boundary layer. Thus, the relatively larger size of cells is used and computational burden can be significantly alleviated.

In this work, a condensation model is presented. In the proposed model the convective heat transfer coefficient near the cooling plate is evaluated from the standard wall function and it is converted into the mass transfer coefficient with the aid of the heat and mass transfer analogy. By using the diffusion theory and the derived mass transfer coefficient, the condensation rate is determined. Finally, the proposed model is implemented into the CUPID code and validated in comparison with the COPAIN experiments [5].

2. Condensation model

The temperature profile based on the standard wall function is given as follows [6]:

$$T^+ \equiv \frac{(T_w - T_P)\rho c_p u^*}{q''} = \Pr_t \left[\frac{1}{\kappa} \ln\left(E y^*\right) + P \right] \qquad (1)$$

where T_w refers to the wall temperature, T_P the temperature at wall adjacent cell, ρ the density, c_p the specific heat, u^* the friction velocity, q'' the heat flux, $\Pr_t$ the turbulent Prandtl number given by 0.9, κ the von Karman constant given by 0.42, E the empirical constant given by 9.793, y^* the dimensionless wall distance and P is computed as

$$P = 9.24\left[\left(\frac{\Pr}{\Pr_t}\right)^{3/4} - 1\right]\left[1 + 0.28\exp\left(-0.007\frac{\Pr}{\Pr_t}\right)\right] \qquad (2)$$

where Pr denotes the Prandtl number. Note that the above equation (1) is valid for $y^* > 11.225$. From the definition of the heat transfer coefficient h, Eq. (1) can be modified to as:

$$h = \frac{(T_w - T_p)\rho c_p u^*}{(T_w - T_{ref})\Pr_t\left[\frac{1}{\kappa}\ln\left(E y^*\right) + P\right]} \qquad (3)$$

Here, T_{ref} represents the reference temperature determined by an experimenter. Using heat and mass transfer analogy, the Nusselt number can be replaced by the Sherwood number. Thus, the mass transfer coefficient can be expressed as follows:

$$h_m = h\left(\frac{\rho D}{k}\right)\left(\frac{Sc}{\mathrm{Pr}}\right)^{1/3} \tag{4}$$

where D is the binary diffusion coefficient defined as

$$D = 2.6\times10^{-5}\left(\frac{T_g}{T_o}\right)^{1.75}\left(\frac{P_o}{P}\right) \tag{5}$$

Here, the reference temperature and pressure are $T_o = 298\ \mathrm{K}$ and $P_o = 100{,}000\ \mathrm{Pa}$, respectively. In Eq. (4), Sc represents the Schmidt number, which is a counterpart of the Prandtl number in heat transfer.

The condensation rate is governed by the diffusion rate of steam towards the liquid film interface. The mass flux of steam is composed of two contributions, i.e., a contribution due to the diffusion and a contribution due to the advection as follows:

$$m_v'' = -\rho D\left.\frac{\partial W_v}{\partial y}\right|_i + W_{v,i}\left(m_v'' + m_{nc}''\right) \tag{6}$$

where W_v refers to the mass fraction of steam, $W_{v,i}$ that of steam at the interface, y the normal direction to the wall, m_{nc}'' the mass flux of non-condensable gas.

Since the non-condensable gas is not permeable to the condensate film, Eq. (6) can be simplified to as:

$$m_v'' = \frac{-\rho D\left.\frac{\partial W_v}{\partial y}\right|_i}{1-W_{v,i}} = h_m\frac{W_{v,\infty}-W_{v,i}}{1-W_{v,i}} \tag{7}$$

By introducing the Bird's suction factor into Eq. (7), we obtain the following expression for the mass flux:

$$m_v'' = -h_m\frac{W_{v,\infty}-W_{v,i}}{1-W_{v,i}}\frac{\ln(1+B)}{B} = -h_m\ln\left(\frac{1-W_{v,i}}{1-W_{v,\infty}}\right) \tag{8}$$

where B is the Spalding mass transfer number given by

$$B = \frac{W_{v,i}-W_{v,\infty}}{1-W_{v,i}} \tag{9}$$

The above equation (8) indicates that the mass flux is determined by steam concentration difference between the bulk and the interface for a given mass transfer coefficient. The interfacial properties are evaluated by iterative calculations in such a way that the heat balance between the film and the bulk gaseous mixture are satisfied. Using the Nusselt's theory the film thickness is estimated to be the order of several micro meter and the film resistance is believed to be negligible for the containment thermal-hydraulic applications where non-condensable concentration is typically high. For practical purpose, thus, this work assumes that the film resistance is negligible and interfacial properties can be approximated to those for the wall temperature. By applying this assumption, Eq. (8) can be further simplified to as:

$$m_v'' = -h_m\ln\left(\frac{1-W_{v,w}}{1-W_{v,\infty}}\right) \tag{10}$$

where the mass fraction of water vapor at the wall $W_{v,w}$ is calculated as

$$W_{v,w} = \frac{X_{v,w}M_v}{X_{v,w}M_v + \left(1-X_{v,w}\right)M_{nc}} \tag{11}$$

Here, M_v and M_{nc} are molecular weights of water vapor and non-condensable gas, respectively, and $X_{v,w}$ is the steam mole fraction calculated from the saturation pressure for the wall temperature.

Finally, the condensation mass flux is implemented into the wall source terms in the CUPID code as follows:

$$\Gamma_w = -h_m\ln\left(\frac{1-W_{s,w}}{1-W_{s,b}}\right)\frac{A_{cell}}{V_{cell}} \tag{12}$$

where A_{cell} and V_{cell} are the surface area and the volume of the wall adjacent cells, respectively.

3. Validation of Condensation model

In this section, the proposed condensation model based on a turbulent wall function and HMTA is validated by comparing with the COPAIN experiments and other commercial code results.

The COPAIN experiments were conducted to investigate the steam condensation on the vertical wall in the presence of non-condensable gas under forced convection [6]. The test section is mainly composed of a rectangular flow channel and a vertical cooling plate. The cross-sectional area of the channel is 0.6 m × 0.5 m and the height is 2.5 m. The cooling plate with 0.6 m in width and 2 m in height is installed on the side wall of the channel and the back side of the cooling plate is cooled by water at constant temperature.

For validations, four different cases are considered as given in Table 1. In each case, the pressure is about 1 bar. The mass fractions of the non-condensable gas are comparable. The inlet velocities range from 0.3 to 3.0 m/s.

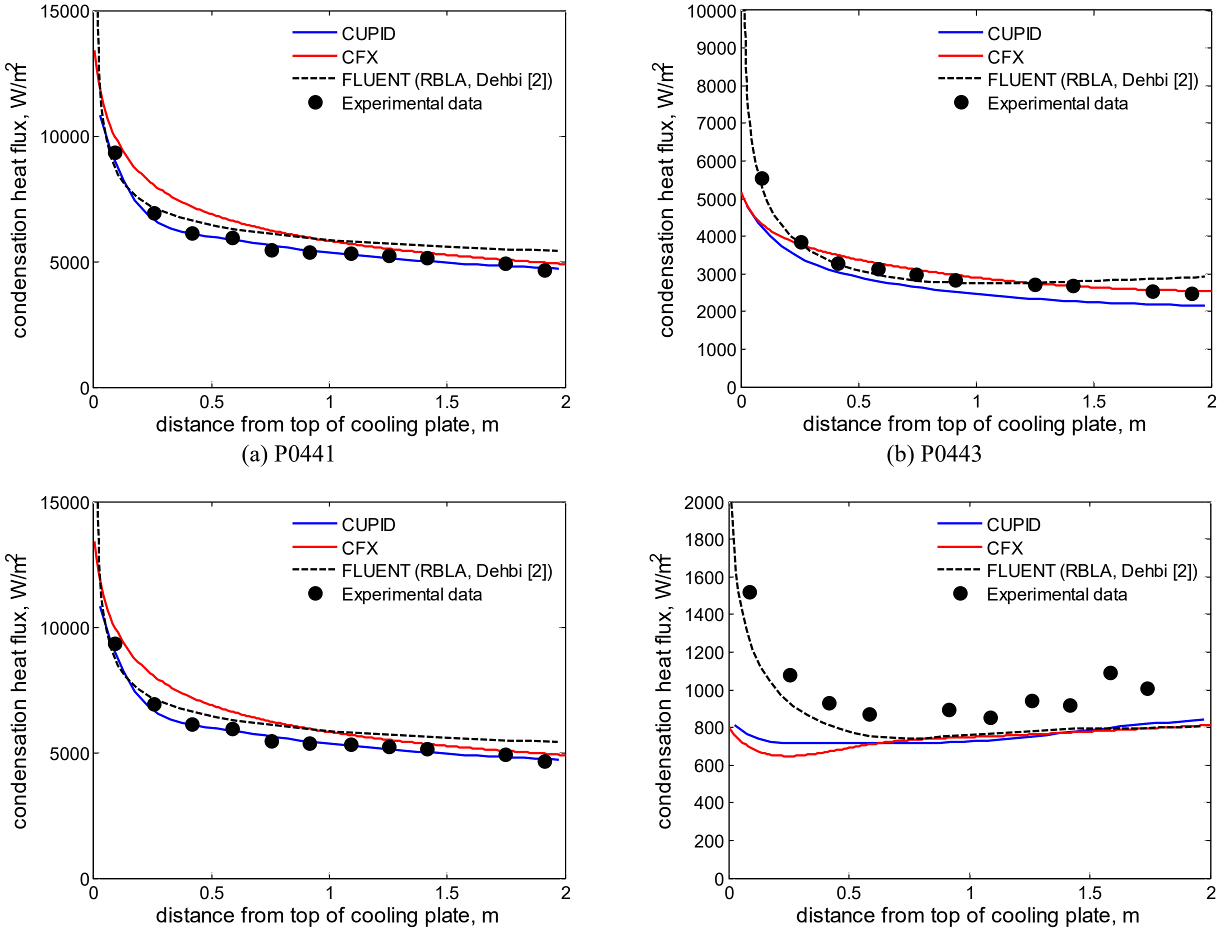
Table 1. Conditions for simulations

Cases	U, m/s	W_{nc}	P, bar	T_{in}, K	T_w, K
P0441	3.0	0.767	1.02	353.2	307.4
P0443	1.0	0.772	1.02	352.3	300.1
P0444	0.5	0.773	1.02	351.5	297.7
P0344	0.3	0.864	1.21	344.4	322.0

Figure 1 shows the comparison of the wall heat fluxes. It is observed from both computational and experimental results that the wall heat flux increases with the mixture velocity, indicating that higher velocities result in higher condensation rate. This would be explained by the fact that convection heat transfer increases with the velocity and consequently this leads to increase in condensation heat transfer from the heat and mass transfer analogy.

In the developing region of the boundary layer, the CUPID code tends to show substantial under-predictions compared to experimental data and other code results. As the flow becomes developed, the deviation is significantly reduced and even CUPID code gives comparable predictions to those of the resolved boundary layer approach and the experiments.

These results were drawn by considering the steam condensation over a simple flat plate. For more complicated geometry involving the 3D phenomena, the proposed model would be further tested. Also, the adequacy of the resolved boundary approach would be investigated for future work.



(a) P0441 (b) P0443

(c) P0444 (d) P0344

Figure 1. Comparison of CUPID code predictions with experimental data and other commercial codes

4. Conclusions

In this work, a steam condensation model based on a turbulent wall function and the heat and mass transfer analogy was presented and it was implemented to the CUPID code. The validation results showed that the proposed model was overall good agreements with experimental data and other codes in developed flow region. However, in entrance region some deviations were observed. For future work, the proposed condensation model would be further tested for complicated geometry involving 3D phenomena and the adequacy of the resolved boundary approach would be investigated.

REFERENCES

[1] M. Houkema, N. B. Siccama, J. A. Lycklama à Nijeholt, and E. M. J. Komen, Validation of the CFX4 CFD code for containment thermal-hydraulics, Nuclear Engineering and Design, Vol. 238, p. 590, 2008.
[2] A. Dehbi, On the adequacy of wall functions to predict condensation rates from steam-noncondensable gas mixtures, Nuclear Engineering and Design, Vol. 265, p. 25, 2013.
[3] L. Vyskocil, J. Schmid, and J. Macek, CFD simulation of air-steam flow with condensation, Nuclear Engineering and Design, Vol. 279, p. 147, 2014.
[4] G. Zschaeck, T. Frank, and A. D. Burn, CFD modelling and validation of wall condensation in the presence of non-condensable gases, Nuclear Engineering and Design, Vol. 279, p. 137, 2014.
[5] X. Cheng, P. Bazin, P. Cornet, D. Hittner, J. D. Jackson, J. Lopez Jimenez, A. Naviglio, F. Oriolo, and H. Petzold, Experimental data base for containment thermal-hydraulic analysis, Nuclear Engineering and Design, Vol. 204, p. 267, 2001.
[6] B. E. Launder and D. B. Spalding, The numerical computation of turbulent flows, Computer Methods in Applied Mechanics and Engineerings, Vol. 3, p. 269, 1974.