Measurements of Temperature Distributions and Condensation Heat Fluxes for Downward Flows of Steam-Air Mixture in a Circular Pipe*

Michio MURASE** Yoichi UTANOHARA** Raito GODA***
Takeyuki SHIMAMURA*** Shigeo HOSOKAWA***† Akio TOMIYAMA***

Abstract Three quantities were measured to provide a database for validation of computations with a CFD (computational fluid dynamics) code. The quantities were the radial temperature distributions of steam-air mixture in a vertical circular pipe (diameter, 49.5 mm), the temperature gradients in the pipe wall (thickness, 5.5 mm) and the radial temperature distributions of the cooling water in the annulus gap (8.25 mm) outside the pipe. From these temperature distributions, three kinds of condensation heat fluxes were obtained from the enthalpy decreasing rate of the steam-air mixture based on the assumption of saturated conditions, the temperature gradient in the pipe wall, and the enthalpy increasing rate of the cooling water. These three heat fluxes were different especially in the downstream region, where the heat flux was low, so that the average of the three heat fluxes was used to evaluate factors affecting the condensation heat transfer. This average was expressed by a function of the steam density difference between the main flow of the mixture and the mixture on the wall surface, the average steam velocity and the thermal boundary layer thickness. The suction effect due to condensation was also discussed.

Keywords: condensation, non-condensable gas, circular pipe, temperature distribution, heat flux

1. Introduction

Condensation in the presence of non-condensable gases is the dominant heat and mass transfer phenomenon in the containment vessel (CV) of a light water reactor during the course of a loss-of-coolant accident (LOCA). The structural integrity of the CV under LOCA and severe accident (SA) conditions has been evaluated by various thermal-hydraulic codes (see the review by de la Rosa et al. [1]); many of the codes employ a lumped parameter model, and an empirical correlation or the heat and mass transfer analogy is used for the condensation heat transfer in them. Green and Almenas [2] reviewed primary parameters for determining condensation heat transfer to containment structures, and they showed that previously proposed correlations underestimated or

overestimated the local condensation heat transfer coefficients measured in the Heiss-Dampf Reactor (HDR) containment with a volume of 11060 m³ and a height of 60 m depending on fluid conditions such as local velocity and steam concentration. These discrepancies were mainly due to the difference between applicable conditions of correlations and local conditions in data.

Lumped parameter codes, which have been used to predict thermal-hydraulic phenomena in the CV, have the merit to provide a picture of the expected overall plant behavior with a reasonably limited computational effort. On the other hand, the developments of computational machines and techniques make it possible to address even complicated geometries by use of computational fluid dynamics (CFD) codes. Mimouni et al. [3] and Dehbi

* Received: 15 July 2019/Accepted: 25 September 2019/Published online: 1 November 2019

** Institute of Nuclear Safety System, Inc., 64 Sata, Mihama-cho, Mikata-gun, Fukui-ken, 919-1205 Japan
TEL: +81-(775)-37-9100 FAX: +81-(770)-37-2009 E-mail: murase@inss.co.jp

*** Kobe University †Present affiliation: Kansai University

et al. [4] proposed a CFD model for the steam condensation in the presence of non-condensable gas, and both groups conducted a numerical simulation of TOSQAN experiments [5], where steam was injected into a closed cylindrical vessel (diameter, 1.5 m; height, 4.8 m) through a vertical pipe (diameter, 0.041 m) located centrally in the lower section of the vessel. Shibamoto et al. [6] carried out experiments similar to the TOSQAN experiments by using a test vessel called CIGMA to understand thermal hydraulic phenomena during severe accidents, and Ishigaki et al. [7] conducted a numerical simulation of the experiments taking condensation on the test vessel wall into account. All of them [3,4,7] obtained good agreement between computed results and measured values, but local condensation heat transfer coefficients were not well validated due to lack of local heat transfer data. In the framework of SARNET (Severe Accident Research Network of Excellence), benchmarking activities were done in four steps for the condensation rates on a vertical flat plate with a height of 2 m in a 0.34 x 0.34 m square channel to develop updated CFD models for reactor containment analysis [8]. In these benchmarking activities, CFD codes such as CFX and FLUENT were used, the bulk condensation in the steam-air mixture and the wall condensation on the vertical flat plate were addressed, and good predictive capabilities were achieved for the condensation rates. However, the temperature distributions of the steam-air mixture, which are important for validation of the bulk condensation model, were not reported. Kang and Kim [9] measured temperature and velocity profiles of a steam-air mixture flow over a nearly horizontal flat plate, and they did numerical simulations for the flow. They indicated that the gas mixtures became supersaturated near the interface, the gas mixture temperature profile was greatly concaved toward the interface, and the equilibrium model (where the phase changes at saturation conditions) underestimated the mixture temperature and the heat transfer coefficient, when the temperature difference between the saturated steam-air mixture and the cold wall was large. In the CFD application to predict thermal-hydraulic phenomena in the CV under LOCA

and SA conditions, the bulk condensation in the steam and non-condensable gas mixture and the wall condensation on structural walls should be addressed and the condensation models should be validated against temperature distributions of the steam and non-condensable gas mixture in the thermal boundary layer. However, there are very few published databases for temperature distributions of the steam and non-condensable gas mixture in the thermal boundary layer.

Utanohara et al. [10] did a numerical simulation of flow and temperature fields in the CV of a three-loop PWR (pressurized water reactor) during operation by using the CFD code FLUENT 13; this could be used as the initial conditions for a numerical simulation of thermal-hydraulic phenomena in the CV under LOCA conditions. Utanohara et al. [11] also did a numerical simulation for the experiment [12] of steam injection into a small-scale closed cylindrical vessel filled with air simulating a LOCA by using the CFD code FLUENT 15.0 to make a CFD model of the thermal flow field in the test vessel. This steam injection experiment, however, is not suitable for validation of condensation models because there are many influencing parameters such as temperature rise due to compression, condensation in the mixing boundary layer of the steam jet and ambient air, and a complex distribution of heat transfer coefficients on the vessel wall along the jet impingement and wall jet.

The objective of this study was to provide a database, which can be used to validate the prediction capability for temperature distributions of the steam and non-condensable gas mixture in the thermal boundary layer in CFD models. For this purpose, we measured radial temperature distributions of the steam-air mixture in a simple vertical circular pipe, the temperature gradients in the pipe wall and radial temperature distributions of the cooling water in the annulus gap outside the pipe (the geometry of the test section and test conditions did not simulate thermal-hydraulic features in the CV). From these temperature distributions, we obtained three kind of condensation heat fluxes from the enthalpy decreasing rate of the steam-air mixture based on

Fig. 1 Schematic diagram of experimental apparatus.

the assumption of saturated conditions, the temperature gradient in the pipe wall, and the enthalpy increasing rate of the cooling water. We also evaluated primary parameters affecting the condensation heat fluxes.

2. Experimental Method

Many experiments have been done to measure condensation heat transfer of steam and non-condensable gas mixtures in a circular pipe or a flat plate, and Huang et al. [13] reviewed empirical correlations and theoretical models proposed for condensation heat transfer in the presence of non-condensable gases. In most of the previous studies, one-dimensional temperature distributions along the mixture flow were measured and the proposed correlations or theoretical models were mainly for one-dimensional computations. In this study, therefore, we measured radial temperature distributions of a steam-air mixture in a simple vertical circular pipe to provide a database for validation of computations with a CFD code.

2.1 Experimental Apparatus

The schematic diagram of the experimental apparatus is shown in Fig. 1. The apparatus consisted of a test section, a steam boiler (Miura, ME-20), a steam regulator (TLV, S-COS-16), a steam heater (Nippon Heater, SGA, 2 kW), an air compressor (ANEST IWATA, SLP-1501EB), an air drier (ANEST IWATA, RDG-150C), an air heater (Nippon Heater, SGA-3425, 4 kW), a steam-air mixer (a simple

mixing tee), an inlet section ($L/D = 10$) to the test section, an exhaust pipe, and a cooling water system to feed low-temperature water into the cooling annulus in the test section. The pipes for air after the air heater, steam and steam-air mixture were kept warm with low-power heaters. The steam-air mixture from the test section was exhausted to the atmosphere.

The measurement error was $\pm 2.5\%$ for the air flow meter (Nippon Flowcell, FLT-N) and $\pm 2\%$ for the steam flow meter (Nippon Flowcell, JBH-5-FD). The pressure and temperature of the inlet steam-air mixture were measured in the inlet section. The measurement error was ± 3 kPa for the pressure transducer (Nagano Keiki, KH15-M33) in the pressure range of 0-0.3 MPa and ± 0.5 °C for the T-type sheathed thermocouple (Okazaki MFG, T35, class 1, grounded).

The test section is shown in Fig. 2. The test section consisted of a stainless steel pipe with the diameter of 49.5 mm and the wall thickness of 5.5 mm, a polycarbonate outer pipe forming a cooling water annulus with the gap of 8.25 mm, and the inlet and outlet plenums for cooling water. The length of the cooling region was 0.61 m including the inlet and outlet plenums.

Locations of temperature measurement are shown in Fig. 3. T-type sheathed thermocouples ($\phi 0.5$, flat at the top end) were embedded in the stainless steel pipe at the radial positions of $r = 26.25$ mm (1.5 mm from the inner surface) and 28.25 mm

Fig. 2 Test section.

Fig. 3 Locations of temperature measurement: Gas (U) and (L), thermocouples at 50 mm and 20 mm upstream of the support plates, respectively (see Photo 1).

(2.0 mm from the outer surface) and the axial locations of $z = 10, 50, 70, 90, 140, 240, 390$ and 500 mm from the top of the cooling region to measure the temperatures in the heat transfer wall. The same kind of T-type sheathed thermocouples ($\phi 0.5$, flat at the top end) were inserted from the outside of the test section at $z = 50, 70, 90, 140, 240, 390$ and 500 mm to measure temperatures of the cooling water, and temperatures were measured at $r = 30.25, 30.35, 31.25, 32.25, 34.25$ and 38.25 mm ($y = 0, 0.1, 1, 2, 4$,

Photo 1 Thermocouple (TC) array.

8 mm from the outer surface of the heat transfer pipe) by moving the thermocouples.

2.2 Temperature Measurement of Mixture

The thermocouple array shown in Photo 1 was used to measure temperature distributions of the stream-air mixture. Eleven and ten T-type sheathed thermocouples ($\phi 1.0$) were fixed on the support plates at 50 mm and 20 mm upstream from the support plates, respectively. The thermocouple array was inserted in the test section and was supported at the bottom flange with the O-ring seal. Temperatures at $z = 50, 70, 90, 140, 240, 390$ and 500 mm were measured by moving the thermocouple array. The radial locations of thermocouples were $r = -23, -21.3, -18, -14, -10, 0, 11.5, 16, 18.5, 21.5$ and 24 mm for the 50 mm upstream position and $r = -22.5, -21, -18, -14, -10, 12, 16, 19, 21$ and 24 mm for the 20 mm upstream position. These thermocouple locations were determined from photographs with a scale.

2.3 Experimental Conditions

Table 1 lists the main experimental conditions for eleven cases. The experimental conditions did not simulate thermal-hydraulic features such as velocities and temperatures during LOCA conditions in the CV. Our interest was temperature distributions in the thermal boundary layer. The parameters were the steam flow rate W_s and air flow rate W_a to change the air mass flow ratio $x_{a,in}$. Nos. 3-5 were the base case and were for confirmation of repeatability. Nos. 1 and 2 were for the effect of small $x_{a,in}$. Nos. 6-8 were for the effect of flow rates at the same $x_{a,in}$. Nos. 9-11 were for the effect of large $x_{a,in}$.

After mixing steam and air, the steam partial pressure decreases and the saturated temperature

Table 1 Experimental conditions.

No.	$W_{s,s}$ [g/s]	$W_{s,in}$ [g/s]	W_a [g/s]	$x_{a,in}$ [-]	$T_{g,in}$ [°C]	$T_{cw,in}$ [°C]
1	3.98	3.58	2.0	0.36	97.3	18.5
2	4.4	4.03	4.5	0.53	91.5	12.5
3	5.97	5.77	9.0	0.61	87.3	12.7
4	5.94	5.91	9.0	0.60	87.7	22.5
5	5.86	5.86	9.0	0.61	87.5	28.5
6	4.14	3.94	6.0	0.60	87.7	12.7
7	2.0	1.91	3.0	0.61	87.4	12.7
8	1.07	0.84	1.5	0.64	85.9	11.2
9	1.07	0.82	3.0	0.79	74.6	11.0
10	1.09	0.79	6.0	0.88	61.8	11.2
11	0.94	0.74	9.0	0.92	53.5	11.2

Pressure in the inlet section $P_{in} = 0.124\text{--}0.127$ MPa, Cooling water flow rate $W_{cw} = 56$ g/s, $W_{s,s}$, supplied steam flow rate; $W_{s,in}$, steam flow rate in the inlet section; W_a , air flow rate; $x_{a,in}$, air mass flow ratio ($=W_a/(W_{s,in}+W_a)$); $T_{g,in}$, mixture temperature in the inlet section; $T_{cw,in}$, cooling water inlet temperature

decreases, and so steam becomes superheated. In our experiments, therefore, air at room temperature was mixed with steam to keep the saturated condition after mixing. This means that the steam flow rate in the inlet section $W_{s,in}$ differed from the steam flow rate measured before mixing $W_{s,s}$ due to condensation in the steam-air mixer. We obtained $W_{s,in}$ from the air flow rate W_a , and the pressure P_{in} and mixture temperature $T_{g,in}$ measured in the inlet section assuming a Gibbs-Dalton mixture of steam and an ideal non-condensable gas [14]:

$$W_{s,in}/W_a = M_s P_{s,in} / \{M_a (P_{in} - P_{s,in})\} \quad (1)$$

where M [kg/mol] is the molecular weight. The steam partial pressure in the inlet section $P_{s,in}$ was obtained from $T_{g,in}$ assuming the saturated condition. The air mass flow ratio was in the range of $x_{a,in} = 0.36\text{--}0.92$. The mixture temperature in the inlet section was in the range of $T_{g,in} = 53.5\text{--}97.3$ °C depending on $x_{a,in}$. The Reynolds number of the mixture in the inlet section was in the range of $Re_{g,in} = \rho_g v_{g,in} D / \mu_g = 3600\text{--}25000$. The inlet temperature of cooling water

$T_{cw,in}$ was different depending on the season during which the experiments were done. The Reynolds number of cooling water was about 500, which was selected to obtain a detectable temperature rise and avoid local natural circulations.

3. Temperature Distributions

3.1 Radial Temperature Distributions

Examples of radial temperature distributions are shown in Fig. 4. We assumed circumferentially symmetrical temperature distributions because clear differences were not found between radially plus and minus regions in Fig. 3 (this indicated good mixing of steam and air up to the test section). In No. 4, the condensation heat transfer was high and radial temperature differences were large in the wall and the cooling water. In No. 11, the condensation heat transfer was low, radial temperature differences were small in the wall and the cooling water, and the temperature increase of the cooling water along the flow direction was small.

3.2 Thermal Boundary Layer Thickness

The thermal boundary layer thickness is shown in Fig. 5. The thermal boundary thickness δ_T is defined by the radial location y from the cold wall surface at which the dimensionless temperature difference dT^* is 0.99:

$$\delta_T = y|_{dT^*=0.99} \quad (2)$$

$$dT^* = (T - T_w) / (T_c - T_w) \quad (3)$$

where T_c and T_w are temperatures at the center of the pipe and on the cold wall surface, respectively. For Nos. 3-5 in Table 1, the thermal boundary layer thickness was expressed by $\delta_T/R = 0.20 (z/D)^{0.40} \pm 0.03$, where the uncertainty of ± 0.03 was difference among Nos. 3-5. The δ_T/R for Nos. 2, 6 and 7 was within the uncertainty of Nos. 3-5, but the δ_T/R for No. 1 was thinner than that for Nos. 3-5. In No. 1, the air mass flow ratio $x_{a,in}$ was lowest in the experiments, and condensation heat fluxes were high. The δ_T/R for Nos. 8-11, where the steam flow rate $W_{s,in}$ was low and condensation heat fluxes were low, was thicker than that for Nos. 3-5.

Fig. 4 Radial temperature distributions (Nos. 4 and 11: see Table 1).

Fig. 5 Thermal boundary layer thickness (Nos. 1-11: see Table 1).

Fig. 6 Dimensionless temperature distributions (Nos. 3-5).

Fig. 7 Effects of experimental conditions on dimensionless temperature distributions (symbols: see Fig. 6).

Plotted values of Fig. 5 showed that the δ_T/R became thinner with increasing condensation heat fluxes. This is the well-known “suction effect” [4,15] in which the absorption of steam into the condensate liquid film causes shrinkage of the boundary layer.

3.3 Temperatures in Boundary Layer

Dimensionless temperature distributions in the thermal boundary layer for Nos. 3-5 are shown in Fig. 6. The maximum uncertainties of $d(y/\delta_T) = \pm 0.1$ and $d(dT^*) = \pm 0.006$ corresponded to uncertainties of $dy = \pm 0.5$ mm in the thermocouple locations and $dT = \pm 0.5$ °C in the temperature measurement, respectively.

Temperatures in Fig. 6 were clearly divided into two regions, where temperatures were expressed by a linear function or an exponent function. For the temperatures in the region expressed by a linear function, there were not enough data points and the uncertainties for the measurement locations were large, and we could not discuss this in detail. Most of the temperatures were within the maximum uncertainties. Two low temperature points in No. 5 (in the broken line) might be due to wetting of the thermocouples. In the turbulent boundary layer expressed by an exponent function, the temperature gradient was relatively low. It might be due to the suction effect.

Fig. 7 shows effects of experimental conditions on the dimensionless temperature distribution. Temperature distributions for Nos. 7 and 8 are not shown but they were similar to those for Nos. 6 and 9,

respectively. Temperatures for Nos. 1, 2 and 6 were higher than those for Nos. 3-5. Four high temperature points in No. 1 (in the broken line) might be due to a local superheated state. Temperatures for Nos. 9-11 were lower than those for Nos. 3-5. Results in Fig. 7 showed that the suction effect affected not only the boundary layer thickness but also the temperature distribution in the boundary layer.

4. Condensation Heat Flux

4.1 Evaluation Method

The uncertainty of the condensation heat flux strongly depends on its measurement method. The temperature gradient in the heat transfer wall is widely used to measure the condensation heat flux q_w [16]:

$$q_w = \lambda_w (T_{r1} - T_{r2}) / \{r_{w,in} \ln(r_2/r_1)\} \quad (4)$$

where q_w [kW/m²] is defined on the inner surface of the wall ($r = r_{w,in}$), r_1 and r_2 are thermocouple locations, and λ_w [kW/mK] is the thermal conductivity of the wall.

Nagae et al. [14] obtained the condensation heat flux from the enthalpy increasing rate of the cooling water or the enthalpy decreasing rate of the steam-air mixture based on the assumption of saturated conditions. They used one temperature for the cooling water or the steam-air mixture at the center in a cross section. In this study, we used a radial temperature distribution for the cooling water or the steam-air

mixture in a cross section. The condensation heat flux q_{cw} from the enthalpy increasing rate of the cooling water is expressed by:

$$q_{cw} = -(dQ_{cw}/dz)/2\pi r_{w,in} \quad (5)$$

$$Q_{cw} = \sum c_{p,cw} (v_{cw,i}/v_{cw,ave}) W_{cw} T_{cw,i} \times \{\pi(r_{i+1}^2 - r_i^2)/A_{cw}\}, \quad (6)$$

where A_{cw} [m²] is the flow area, $c_{p,cw}$ [kJ/kg·K] is the specific heat, Q_{cw} [kW] is the thermal energy in a horizontal cross section transferred by the cooling water, $v_{cw,i}$ [m/s] is the local velocity, and $v_{cw,ave}$ [m/s] is the average velocity. The radial location r_i was defined by the center of the neighboring temperature measurement locations. The velocity distribution of a laminar flow was used for $v_{cw,i}$.

Applying Eq. (1) to obtain the local steam mass flux, we express the condensation heat flux q_g from the enthalpy decreasing rate of the steam-air mixture by:

$$q_g = (dQ_g/dz)/2\pi r_{w,in} \quad (7)$$

$$Q_g = \sum [M_s P_{s,i} h_s(T_{g,i})/\{M_a (P_{in} - P_{s,i})\} + h_a(T_{g,i})] \times (v_{g,i}/v_{g,ave}) W_a \{\pi(r_{i+1}^2 - r_i^2)/A_g\}, \quad (8)$$

where A_g [m²] is the flow area, $h_a(T_{g,i})$ and $h_s(T_{g,i})$ [kJ/kg] are the enthalpy of air and the saturated steam at the temperature $T_{g,i}$ respectively, Q_g [kW] is the thermal energy in a horizontal cross section transferred by the steam-air mixture, $v_{g,i}$ [m/s] is the local velocity, and $v_{g,ave}$ [m/s] is the average velocity. The velocity distribution of the mixture, which was obtained from a three-dimensional computation with the CFD code FLUENT 17 and standard k-ε turbulence model, was used for $v_{g,i}$.

Fig. 8 compares heat fluxes obtained from the temperature gradient in the pipe wall q_w , the enthalpy increasing rate of the cooling water q_{cw} , and the enthalpy decreasing rate of the steam-air mixture q_g . q_{cw} and q_g are shown by horizontal lines because they are average values in the region of two neighboring thermocouple locations. All q_w values were larger than the axially averaged heat flux obtained from the total enthalpy increase of the cooling water. This might be due to the heat loss through the thermocouples, because the ratio of the thermocouple

Fig. 8 Axial heat flux distributions.

Fig. 9 Average heat fluxes q_{ave} in Nos. 3-5.

inserted depth to the diameter was 4.0 (= 2 mm/0.5 mm). The q_{cw} value might be most reliable, but it sometimes showed an unreasonable value. This might be due to long measurement time, because the thermocouples to measure the cooling water temperature distribution were manually move in the radial direction for each axial location z . The $q_{g,u}$ and $q_{g,l}$, which were respectively obtained from temperatures with the 50 mm and 20 mm upstream thermocouples (cf. Photo 1), sometimes showed an unreasonable value like the value at about $z = 0.1$ m. The reason for the small heat flux at about $z = 0.1$ m was not clear, but it might be due to wetting of the thermocouples as shown in Figs. 6 and 7 or the periodic change of the steam flow rate caused by the pressure change in the steam boiler because the

thermocouple array was axially moved during the measurement. The $q_{g,u-l}$, which was obtained from the difference of enthalpies at locations of the 50 mm and 20 mm upstream thermocouples, showed relatively reasonable values in No. 4. However, the $q_{g,u-l}$ sometimes showed unreasonable values. As mentioned above, each of the quantities q_w , q_{cw} , q_g , and $q_{g,u-l}$ had technical issues. Therefore, the average value of q_w , q_{cw} , and $q_{g,u}$ was used in this study to mitigate effects of unreasonable values (this selection was made from an engineering point of view). $q_{g,u}$ with the same measuring location as q_{cw} was used for q_g ,

$$q_{ave,j} = \{q_{cw,j} + q_{g,u,j} + (q_{w,j-1} + q_{w,j})/2\}/3 \quad (9)$$

Fig. 9 shows the average heat fluxes q_{ave} in Nos. 3-5. The q_{ave} values in Nos. 3-5 were within $\pm 20\%$. The heat fluxes in this study were larger than heat fluxes obtained with the empirical correlation by Araki et al. [16] computed for No. 4 conditions.

$$h_c [\text{kW/m}^2\text{K}] = 2.11 \times 10^{-4} Re_g^{0.8} (P_a/P_l)^{-0.99}, \quad 2300 \leq Re_g \leq 21000 \quad (10)$$

Re_g is the mixture Reynolds number. Heat fluxes with Eq. (10) increased due to the large temperature difference ($T_{g,c} - T_w$) with increasing z . In the experiments by Araki et al. [16], the ratio of the length L to the diameter D in the cooling section was 40 ($L/D = 2000 \text{ mm}/49.5 \text{ mm}$) and the measured heat transfer coefficients were well expressed by the heat and mass transfer analogy for fully developed flows (which is not shown here). The heat flux in this study agreed with that by Araki et al. [16] at $z = 0.5 \text{ m}$.

4.2 Heat Fluxes

Fig. 10 shows the average heat fluxes q_{ave} in Nos. 1-2 and 6-11. The average heat fluxes in the region of $z = 0-0.5 \text{ m}$ for Nos. 1, 2, 3-5, 6, 7, 8, 9, 10 and 11 were respectively, 47.5, 46.0, 44.0, 42.8, 27.9, 14.0, 11.0, 11.2 and 9.5 kW/m^2 . Shrinkage of the thermal boundary layer due to the suction effect shown in Fig. 5 was in the order of Nos. 1, 2-6, 7, 8 and 9-11. This order was similar to the order of the average heat flux. This showed that the condensation heat flux (i.e. steam mass flux toward the cold wall) affected the shrinkage of the thermal boundary layer.

Fig. 10 Average heat fluxes q_{ave} .

The increase of the q_{ave} in the region of downstream was due to the increase of the temperature difference ($T_c - T_w$).

4.3 Empirical Correlation

Fujii et al. [17] used the ratio of the steam logarithmic mean concentration to the air logarithmic mean concentration $C_{s,ave}/C_{a,ave}$ as a major factor for evaporation from the hot water pool surface and condensation on the cold vertical wall in the presence of air. The $C_{s,ave}/C_{a,ave}$ gave relatively good results for Nos. 1-7 but it was not effective for Nos. 8-11. Due to the suction effect discussed in sections 3.2 and 4.2, we cannot apply the heat and mass transfer analogy without the suction effect. Therefore, the measured boundary thickness δ_T/R was used as one of the effective factors to derive an empirical correlation of the condensation heat flux q_{ave} . Based on the

Fig. 11 Comparison of $q_{ave,cal}$ calculated by Eq. (11) with $q_{ave,m}$ measured.

examinations, we used the gradient of the steam density $(\rho_{s,c} - \rho_{s,w})/(\delta_T/D)$ and the average steam velocity $v_{s,ave}$ as major factors and we derived the following empirical correlation:

$$q_{ave} = 0.0044 \{(\rho_{s,c} - \rho_{s,w})/(\delta_T/D)\} v_{s,ave} h_{fg},$$

$$3200 \leq Re_{g,in} \leq 25000, 0.36 \leq x_{a,in} \leq 0.92 \quad (11)$$

where ρ_s [kg/m³] is the steam density and h_{fg} [kJ/kg] is the latent heat of evaporation or condensation. The coefficient of 0.0044 was obtained to fit heat flux data. In Eq. (11), $\rho_s v_s$ shows the steam mass flux, and the effect of the air flow rate was small.

Fig. 11 compares the $q_{ave,cal}$ calculated by Eq. (11) with the $q_{ave,m}$ measured. The uncertainty of Eq. (11) was $\pm 40\%$. Eq. (11) agreed well with data for Nos. 8-11 in the region of small q_{ave} , but underestimated q_{ave} for No. 1 and overestimated q_{ave} for Nos. 3-5 in the region of large q_{ave} due to the small effect of the air flow rate in Eq. (11).

5. Discussion

5.1 Suction Effect

As discussed in section 4.2, the average heat flux affected shrinkage of the thermal boundary layer due to the suction effect. The suction effect with suction through the pipe wall from single phase turbulent flows was reported by Aggarwal and Hollingsworth [18] and Yeroshenko et al. [19], but we

Fig. 12 Effects of heat fluxes on temperature distributions in the boundary layer.

could find no reports of suction effect data under condensation in a vertical pipe (i.e. temperature distributions in a turbulent boundary layer).

The suction effect affected not only the thermal boundary layer thickness but also the temperature distribution in the turbulent boundary layer as shown in Fig. 12. The exponential value for the dimensionless temperatures decreased with increasing the condensation heat flux. The slope in the region of a linear function should increase with increasing the condensation heat flux, but this was not detected by the thermocouple array shown in Photo 1. This might be possible to measure by traversing a thermocouple in the thermal boundary layer. Fig. 12 will provide a database for validation of numerical simulations with a CFD code on the suction effect under condensation conditions.

5.2 Condensation Heat Flux

The main objective of this experiment was to provide a database for validation of numerical simulations. Therefore, we discussed important behavior to be validated like the suction effect and the heat flux as one of the boundary conditions in numerical simulations.

In numerical simulations, the boundary conditions on the wall surface are generally given by the distribution of the temperatures, heat fluxes, or heat transfer coefficients. In this study, the thermal

resistance of the condensate film was not separately evaluated and condensation heat transfer coefficients were not evaluated. In numerical simulations for the experiments, the heat flux distribution shown in Figs. 9 and 10 can be used.

6. Conclusions

We measured radial temperature distributions of steam-air mixture in a vertical circular pipe with the diameter of 49.5 mm, and the temperature distributions in the thermal boundary layer and condensation heat fluxes were evaluated. The main findings of the present study are as follows:

- (1) The suction effect due to condensation, which shrinks the thermal boundary layer, was observed, and the thermal boundary thickness decreased with increasing the condensation heat fluxes.
- (2) The gradient of the dimensionless temperature difference in the turbulent boundary layer decreased due to the suction effect with increasing the condensation heat fluxes.
- (3) The condensation heat fluxes were in proportion to the steam density difference between the main flow of the mixture and the mixture on the wall surface, the average steam velocity and the inverse of the thermal boundary layer thickness.

Nomenclature

A	: flow area	[m ²]
c_p	: specific heat	[kJ/kg K]
D	: diameter of test section	[m]
dT^*	: dimensionless temperature difference	[-]
h	: enthalpy	[kJ/kg]
h_c	: heat transfer coefficient	[kW/m ² K]
h_{fg}	: latent heat of condensation	[kJ/kg]
L	: pipe length	[m]
M	: molecular weight	[kg/mol]
P	: pressure	[MPa]
Q	: thermal energy	[kW]
q	: heat flux	[kW/m ²]
R	: radius of test section	[m]
Re	: Reynolds number	[-]
r	: radial coordinate	[m]

T	: temperature	[°C]
v	: velocity	[m/s]
W	: mas flow rate	[kg/s]
x_a	: air mass flow ratio	[-]
y	: distance from wall surface	[m]
z	: axial coordinate	[m]

Greek letters

δ_T	: thermal boundary layer thickness	[m]
λ_w	: thermal conductivity of wall	[kW/m K]
μ	: viscosity	[Pa s]
ρ	: density	[kg/m ³]

Subscripts

a	: air
ave	: average
c	: center
cal	: calculation
cw	: cooling water
g	: steam-air mixture
i	: number of radial location
in	: inlet or inner surface
j	: number of axial location
m	: measured
s	: steam
t	: total
w	: wall

References

- [1] de la Rosa, J. C., Escriva, A., Herranz, L. E., Cicero, T. and Munoz-Cobo, J. L., Review on Condensation on the Containment Structure, Progress in Nuclear Energy, Vol. 51, 32-66 (2009).
- [2] Green, J. and Almenas, K., An Overview of the Primary Parameters and Methods for Determining Condensation Heat Transfer to Containment Structures, Nuclear Safety, Vol. 37(1), 26-48 (1996).
- [3] Mimouni, S., Foissac, A. and Lavieville, J., CFD Modeling of Wall Steam Condensation by a Two-Phase Flow Approach, Nuclear Engineering and Design, Vol. 241, 4445-4455 (2011).
- [4] Dehbi, A., Janasz, F. and Bell, B., Prediction of Steam Condensation in the Presence of Noncondensable Gases using a CFD-Based Approach, Nuclear Engineering and Design,

- Vol. 258, 199-210 (2013).
- [5] Vendel, J., Malet, J. and Bentaib, A., Conclusions of the ISP-47 Containment Thermal-Hydraulics, Proc. the 12th Int. Topical Meeting on Nuclear Thermal-Hydraulics (NURETH-12), Paper No. 031 (2007).
 - [6] Sibamoto, Y., Ishigaki, M., Abe, S. and Yonomoto, T., Experimental Study on Outer Surface Cooling of Containment Vessel by Using CIGMA, Proc. the 17th Int. Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-17), Paper No. 21591 (2017).
 - [7] Ishigaki, M., Abe, S., Sibamoto, Y. and Yonomoto, T., CFD Simulation of a CIGMA Experiment CC-PL-04 on the Containment Thermal Hydraulics Affected by the Outer Surface Cooling, Proc. the 10th Japan-Korea Symposium on Nuclear Thermal Hydraulics and Safety (NTHAS10), Paper No. N10P1153 (2016).
 - [8] Ambrosini, W., Forgione, N., Merli, F., Oriolo, F., Paci, S., Kljenak, I., Kostka, P., Vyskocil, L., Travis, J. R., Lehmkuhl, J., Kelm, S., Chin, Y.-S. and Bucci, M., Lesson Learned from the SARNET Wall Condensation Benchmarks, Annals of Nuclear Energy, Vol. 74, 153-164 (2014).
 - [9] Kang, H. C. and Kim, M. H., Characteristics of Film Condensation of Supersaturated Steam-Air Mixture on a Flat Plate, Int. J. Multiphase Flow, Vol. 25, 1601-1618 (1999).
 - [10] Utanohara, Y., Murase, M., Masui, A., Inomata, R. and Kamiya, Y., Numerical Simulation of Temperature Distribution in a Containment Vessel of an Operating PWR Plant, J. of Nuclear Engineering and Radiation Science, Vol. 1, 011002, 12 pages (2015).
 - [11] Utanohara, Y., Murase, M. and Tomiyama, A., Numerical Simulation of Steam Injection Experiment into a Closed Vessel, Proc. 12th Int. Topical Meeting on Nuclear Reactor Thermal-Hydraulics, Operation and Safety (NUTHOS-12), Paper No. 950 (2018).
 - [12] Nozue, T., Fujii, Y., Hayashi, K., Hosokawa, S., Tomiyama, A., Utanohara, Y., Yamamoto, Y. and Murase, M., Validation of Simulation Model for Temperature in a Containment Vessel during Loss-of-Coolant Accident (2) Experiments on Air or Steam Injection, 2015 Annual Meeting of Atomic Energy Society of Japan (in Japanese), H56 (2015).
 - [13] Huang, J., Zhang, J. and Wang, L., Review of Vapor Condensation Heat and Mass Transfer in the Presence of Non-condensable Gas, Applied Thermal Engineering, Vol. 89, 469-484 (2015).
 - [14] Nagae, T., Murase, M., Wu, T. and Vierow, K., Evaluation of Reflux Condensation Heat Transfer of Steam-Air Mixtures under Gas-Liquid Countercurrent Flow in a Vertical Tube, J. Nuclear Science and Technology, Vol. 42(1), 50-57 (2005).
 - [15] Herranz, L. E., Anderson, M. H., and Corradini, M. L., A Diffusion Layer Model for Steam Condensation within the AP600 Containment, Nuclear Engineering and Design, Vol. 183, 133-150 (1998).
 - [16] Araki, H., Kataoka, Y. and Murase, M., Measurement of Condensation Heat Transfer Coefficient inside a Vertical Tube in the Presence of Noncondensable Gas, J. Nuclear Science and Technology, Vol. 32(6), 517-526 (1995).
 - [17] Fujii, T., Kataoka, Y. and Murase, M., Evaporation and Condensation Heat Transfer in a Suppression Chamber of the Water Wall Type Passive Containment System, J. Nuclear Science and Technology, Vol. 33(5), 374-380 (1996).
 - [18] Aggarwal, J. K. and Hollingsworth, M. A., Heat Transfer for Turbulent Flow with Suction in a Porous Tube, Int. J. Heat and Mass Transfer, Vol. 16, 591-609 (1973).
 - [19] Yeroshenko, V. M., Yershov, A. V. and Zaichik, L. I., Heat Transfer for Turbulent Flow in a Circular Tube with Suction or Injection, Int. J. Heat and Mass Transfer, Vol. 27 (8), 1197-1203 (1984).