Experimental Validation for Heat Removal Rate of Single Fin-Tube Heat Exchanger

Kihwan Kim, Kil-Won Park, Tae-Soon Kwon*
Korea Atomic Energy Research Institute, Daedeok-daero 989-111, Yuseong-Gu, Daejeon 305-353, Republic of Korea
**Corresponding author: tskwon@kaeri.re.kr*

1. Introduction

After the Fukushima Daiich nuclear disaster, application of passive system have been a critical topic as an alternatives to mitigate severe accident. The passive auxiliary feed-water system (PAFS) is one of the advanced safety features adopted in the Advanced Power Reactor Plus (APR+) of KOREA, but the cooling capacity of the PAFS was limited with a duration of 8 hours because of a capacity of passive condensate cooling tank (PCCT) used an ultimate heat sink. Recently, a new passive safety system withstanding a station black out (SBO) for more than 72 hours is needed, and thus Korea Atomic Energy Research Institute (KAERI) has been developed a new Hybrid Air Cooling Passive Residual Heat Removal System to extend the cooling capacity of the PAFS.

When the PAFS transfers the decay heat to the PCCT under a postulated accident, the coolant in the PCCT was heated and finally evaporated continuously because of the high surface temperature of PAFS. Therefore, the large amount of steam may be condensate if the steam can be cooled by passing through the fin-tube bundle. Therefore, the inclined circular fin-tube heat exchanger plays an important role in the development of the new passive system, since the overall performance of HAC PRHRS is determined by the cooling capability of each fin-tube.

KAERI constructed a steam condensation test facility to focus on the evaluation of the overall performance of fin-tube heat exchanger. The schematic of the test facility is shown in Fig. 1. The test facility was designed to enable heat transfer experiments under the natural/forced convection conditions. The main test section is the 3m fin-tube bundle with a gradient of 3 degree, and the total number of fin-tube was 13. Figure 2 shows the schematic of circular fin-tube and lay-out of the fin-tube bundle. The coolability of each fin-tube was quantified with the amount of condensation measured by using a load cell at the outlet of fin tube. For validation the experimental test results, the real geometries of test facility and fin-tube were also exactly simulated with the commercial computational fluid dynamics software, ANSYS CFX (Ver. 15.0.7). The condensation rate was calculated with CFX modelling.

The ultimate objective of this study is to design the whole system of HAC PRHRS based on the evaluation of performance of fin-tube heat exchanger. In this paper, some part of study including experiments and numerical method were introduced for a single fin-tube heat exchanger. The details of the experiments are omitted, but we emphasize the experimental validation results from comparison with the numerical test results.

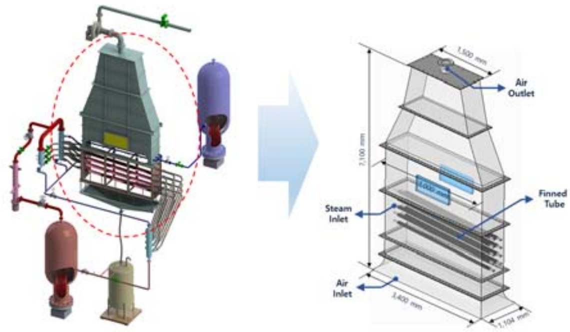

Fig. 1. Schematic of the test facility and CAD Modeling

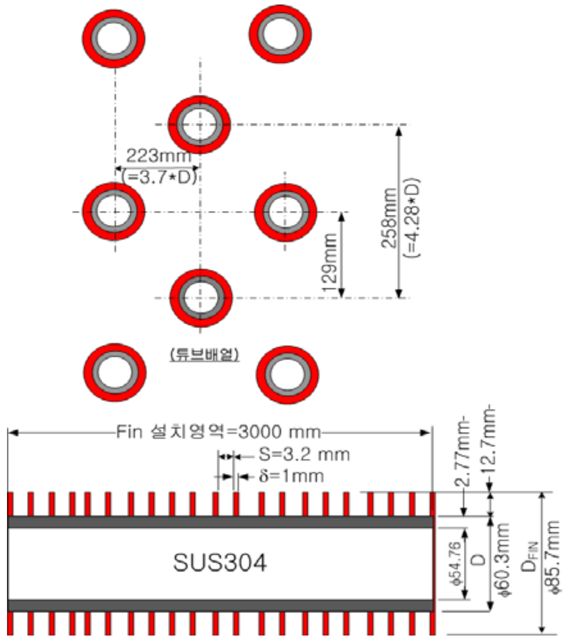

Fig. 2. Schematic of inclined circular fin tube

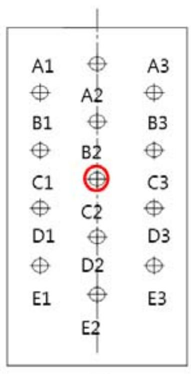

Fig. 3. Location of the single heated fin-tube

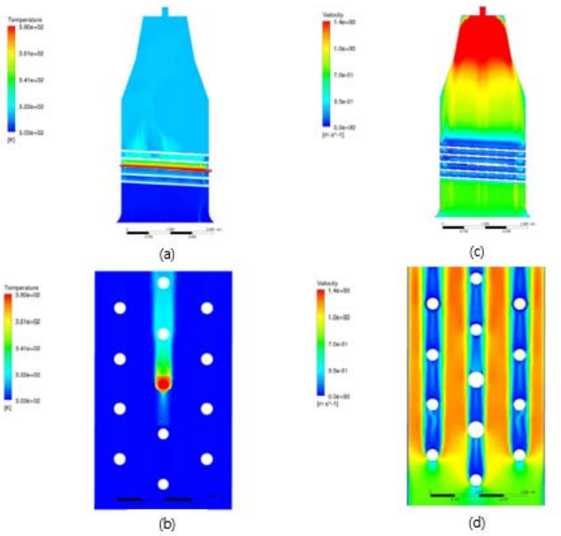

Fig. 4. Temperature (a, b) and velocity distribution (c, d) for V_{air}=0.745m/s

2. Experimental Validation with CFD Codes

The basic experiments were carried out for a single fin-tube heat exchanger since the test results can be used to validate the effect of the flow on the heat removal rate. The location of the single fin-tube was shown in Fig 3, and the test conditions were shown in Table 1, respectively. In this experiments, the condensation rate of fin-tube were measured varying the secondary air side velocity since the condensation rates reveals directly the heat removal rate.

To validate the heat removal rate with a CFD code, the main test section was exactly simulated. The total number of grids for the fin-tube and whole domain were 1.0×10^7 and 5.0×10^7, respectively. Steam and air used as the working fluids, and the k-ω shear stress models were used to simulate the real velocity profile between the viscous sub-layer and buffer-layer. The mass, momentum and energy equations were used and the thermal properties depending on the temperature such as the density, viscosity, thermal capacity, and thermal conductivity were considered to simulate the buoyance effect. Basically, CFX wall condensation model were used to calculate the condensate rate of fin-tube. More detailed descriptions for the applied models can be found in the ANSYS CFX user guide. The solution was considered 'converged' when the residuals of the variables were below 1.0×10^5 and a high resolution scheme was used for the convection-terms-of-momentum equations. Figure 4 shows the temperature and velocity contours as an example of results, and figure 5 shows the condensation rate of fin-tube.

When the velocity is low, the relative error was maximum, but the mass fraction was about 1.7% of inlet steam mass flow rate. The relative error was in the range of 1.9-40.1 %, and the discrepancy may result from the fluid flow phenomena simulation using the k-ω SST turbulence model since the flow region on the adjacent region of fin-tube may be similar to the laminar flow.

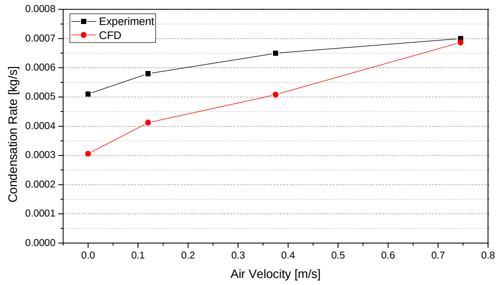

Fig. 5. Comparison of the condensation rate at the outlet of fin-tube

Table 1. Summary of test condition

Air				Steam		
Inlet Velocity [m/s]	Inlet Pressure [bar]	Inlet Temp. [°C]	Outlet Velocity [m/s]	Inlet Temp. [°C]	Inlet Mass Flow Rate [kg/s]	Outlet Pressure [bar]
0	1.0	29.5	-	114.3	0.0118	1.1
0.120	1.0	29.5	6.98	113.7	0.0118	1.1
0.375	1.0	29.5	21.83	113.7	0.0114	1.1
0.745	1.0	29.5	43.36	113.8	0.0115	1.1

3. Conclusion

In this study, the heat removal rate of a slightly inclined single circular fin tube was quantified experimentally varying the air velocity. From the test results, the condensation rate was validated with the commercial CFD software, ANSYS CFX. There were some discrepancies but the predictions were more accurately as increasing the inlet air velocity, named turbulent region, considering the inherent limitation of modelling. This test results are expected to support the design of new PRHRS using CFD code.

Acknowledgments

This research has been performed as a part of the nuclear R&D program supported by the Ministry of Trade, Industry & Energy of the Korean government.