Preliminary analysis on the mixed convection phenomena in the scaled-down VHTR RCCS riser experiment

Dong-Ho Shin [a], Sin-Yeob Kim [a], Chan-Soo Kim [b], Hyoung Kyu Cho [a], Goon-Cherl Park [a*]
[a] *Dept. of nuclear engineering, Seoul National Univ., 1 Gwanak-ro, Gwanak-gu, Seoul 08826*
[b] *Nuclear Hydrogen Reactor Technology Division, Korea Atomic Energy Research Institute, 111 Daedeok-daero 989beon-gil, Yuseong-gu, Daejeon 34057*
[*] *Corresponding author: parkgc@snu.ac.kr*

1. Introduction

Very High Temperature gas-cooled Reactor (VHTR) that is a concept of GEN-IV reactor has been developed for the application in hydrogen production. [1] VHTR is characterized by high operating temperature and stepped-up inherent safety given by passive safety systems such as Reactor Cavity Cooling System (RCCS). RCCS significantly serves the inherent safety of VHTR by cooling the decay heat from the reactor vessel. The performance of RCCS is the key parameter to ensure the safety of the reactor. And it depends on the heat removal rate by the RCCS riser which is a kind of heat exchanger of RCCS.

Several researches on the performance of RCCS have been being conducted and several reduced-scale experiment facilities were constructed for the researches; Korea Atomic Energy Research Institute (KAERI) constructed the quarter-scale RCCS experiment facility, NACEF [2] and Argonne National Laboratory (ANL) constructed half-scale experiment facility, NSTF [3]. University of Wisconsin (UW) also has the quarter-scale experiment facility [4] and Texas A&M University has conducted separate effect test on the jet effect at the upper plenum of RCCS [5]. However, the heat transfer mechanism in the prototype RCCS riser is guessed to be different from that in the scaled-down riser of experimental facility. Without the analysis on the heat transfer phenomena in those conditions, it is impossible to verify the performance of RCCS from the experiment.

In this study, the preliminary analysis on the heat transfer phenomena inside the RCCS riser of the prototype and scaled-down experiments was conducted. The subjects of the analysis were the prototype RCCS risers of PMR 200 [6], and three experimental facilities – NACEF, NSTF and UW experiment facility. Frist, the heat transfer regime was identified using the existing heat transfer regime map and non-dimensional parameters. And then, the CFD calculation was conducted for the conduction effect of the riser duct. Finally, the planned activities based on this analysis were introduced.

2. Preliminary analysis on the heat transfer phenomena in the RCCS riser

2.1 Geometry and boundary conditions of the analysis

The heated length of prototype RCCS riser is 15.5m [7] and its inner width and depth are 240mm and 40mm, respectively. The heated length of NACEF and UW experimental facility was reduced to quarter scale and that of NSTF was reduced to half scale with similar cross-sectional area. The detail geometry information is summarized in Tables 1.

Table 1: Geometry information of each experimental facility

	ANL (NSTF)	KAERI (NACEF)	UW
Total riser height (m)	7.5	4.5	3.76
Heated length of riser (m)	6.82	4.05	3.51
Number ducts (#)	12	6	6
Area heated plate (m^2)	8.820	2.600	3.030

An expected heat imposed on the riser surface of the prototype is about 1098 W which could heat the flow by 98 ºC from the inlet to outlet. Based on this prototype condition, the experimental conditions for three facilities were deduced. Since the scale of experimental facility was reduced, the experimental conditions were also adjusted using two scaling methods; one is the method that was used to design NACEF where the plank number was conserved. [7] The other method [8] is to design and to analyze NSTF and it conserves the Richardson number as a scaling parameter. Depending on the scaling method and its geometry, different conditions such as mass flow rate and heat flux were imposed. In this paper, the heat transfer phenomena inside the riser under those thermal and hydraulic conditions summarized in Table 2 were analyzed.

Table 2: Thermal and hydraulic conditions of each facility

	ANL (NSTF)	**KAERI (NACEF)**	**UW**
Case 1 - Ri_R=1.0			
Mass flow rate per 1 duct (kg/s)	0.030	0.023	0.021
Mass flow rate, system (kg/s)	0.359	0.138	0.129
Mass flow rate, system (kg/hr)	1291.5	497.6	463.3
Q, system (W)	35473.8	13668.3	12724.5
Heat flux on heated surface (kW/m^2)	4.022	5.257	4.199
Case 2 - Pl_R=1.0			
Mass flow rate per 1 duct (kg/s)	0.019	0.011	0.010
Mass flow rate, system (kg/s)	0.227	0.067	0.058
Mass flow rate, system (kg/hr)	818.0	242.9	210.5
Q, system (W)	22468.6	6671.4	5781.9
Heat flux on heated surface (kW/m^2)	2.547	2.566	1.908

2.2 Identification of heat transfer regime in the RCCS riser

In the convective heat transfer, there are three heat transfer modes: forced convection, free convection, and mixed convection. In forced convection, the flow is driven by externally imposed pressure difference. In free convection, on the other hand, the buoyancy force induced by the temperature and density difference is the driving force of the flow. If the flow with the non-uniform density is driven by the pressure difference, the heat transfer mode has significantly modified characteristics from both forced convection and free convection, which is called mixed convection [9]. Depending on the heat transfer mode, the phenomena are so dissimilar that the heat transfer coefficient could also be different.

To obtain the heat transfer correlation used in the scaling analysis of the RCCS riser, it is required to identify whether the correlation is appropriate to the heat transfer regime in the RCCS riser. Based on the flow and thermal conditions inside the riser summarized in Table 2, therefore, the heat transfer mode in it was analyzed using the Metais & Eckert heat transfer mode map [10] which is referred in most of the articles. The parameters for the heat transfer mode map were calculated with inlet and outlet conditions of three experiments (ANL, KAERI, and UW) and the prototype RCCS riser. Heat flux was assumed to be uniform and their hydraulic diameter was used as a characteristic length for the non-dimensional parameter. The parameters were plotted as shown in Fig. 1.

In the pre-analysis, it was revealed that the heat transfer mode would be the forced convection turbulent flow regime for the full scale riser. However, the heat transfer mode for the reduced height test facilities would correspond to the mixed convection turbulent flow regime. Since the heat transfer mode is not identical in the prototype riser and the experiment facilities, a single existing heat transfer correlation cannot be used for the scaling analysis. Moreover, the heat transfer phenomena of the mixed convection are complex due to thermo-physical properties variation and the heat transfer correlation is not established well compared to the forced convection. Therefore, a careful investigation for those is required.

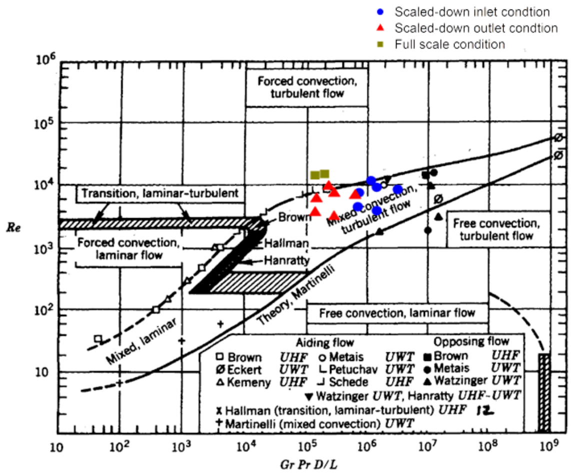

Fig. 1. Heat transfer modes in the prototype riser and the scaled-down experiments

2.3 CFD analysis for the conduction effect of the riser duct

According to KAERI's CFD analysis for the NACEF RCCS test facility, the heat transferred from the heater to the riser duct outer surfaces has highly different distribution depending on the direction of the surfaces. In the KAERI's analysis, six risers and heated wall were modelled, which are identical to the NACEF RCCS test facility. The calculation results for the heat flux on the outer surface of one riser located at the center among six risers were depicted in Fig. 2.

The front surface directly facing the heater has the highest heat flux due to radiation heat transfer and its large view factor and the opposite side has the least one. Since the heat flux is the one of the most important boundary conditions of the SNU experiment, a CFD analysis was conducted for a riser duct with the calculated heat flux. For the simulation, the 4m-long riser duct was modelled as a heat conductor with the carbon steel of 5mm-thickness. The inlet condition was a normal speed of 1.9 m/s in 25 °C and the outlet condition was pressure boundary. In this simulation, it was intended to examine the heat flux distribution on the inner surfaces of the riser duct.

Fig. 2 shows the cross-sectional view of the fluid and structure temperature distribution in the CFD analysis results at the outlet of the duct. From the temperature calculation results, the heat fluxes on each inner wall were analyzed and the averaged heat fluxes on each surface were indicated in the figure. On the outer surfaces, the front surface has the highest heat flux but on the inner faces, the side walls have the largest heat fluxes. This can be explained with the effect of the conduction through the riser duct material and the convective heat transfer coefficient. The fluid temperature distribution shows that the side walls have steeper fluid temperature gradients than the front and rear walls and this means they have higher heat transfer coefficients than the others. Due to this, the heat enters into the front surface moves to the side walls across the riser duct walls and transfers to the fluid on the side surfaces where the heat transfer coefficients are higher than those on the others. From this CFD analysis results, the heat flux boundary conditions for the front, rear and side walls of the SNU experiment were determined. It shows that the inner heat flux ratio of the front wall to the side wall is 1:1.5 normally.

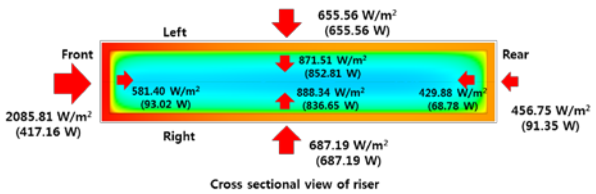

Fig. 2. CFD analysis result: temperature distribution at the outlet cross-section of the riser duct

3. Planned activities

3.1 Experimental facilities for the RCCS riser

To analyze heat transfer phenomena inside the riser, the separate effect experiment for one riser was planned. Since the existing RCCS experimental facilities contain several risers and cavity region, it is difficult to control the heat flux on the riser surface and to obtain detail information such as heat flux and velocity profiles inside the riser. Therefore, the experimental facility was constructed in Seoul National University and the experiment is being prepared. Fig. 3 shows the schematic diagram and picture of SNU experimental facility. The height of riser in the experimental facility is 4m and its width and depth are 240mm and 40mm, respectively, which are same with those of the prototype. A separate heater was attached on each riser surface so that the heat flux at each surface could be controlled independently.

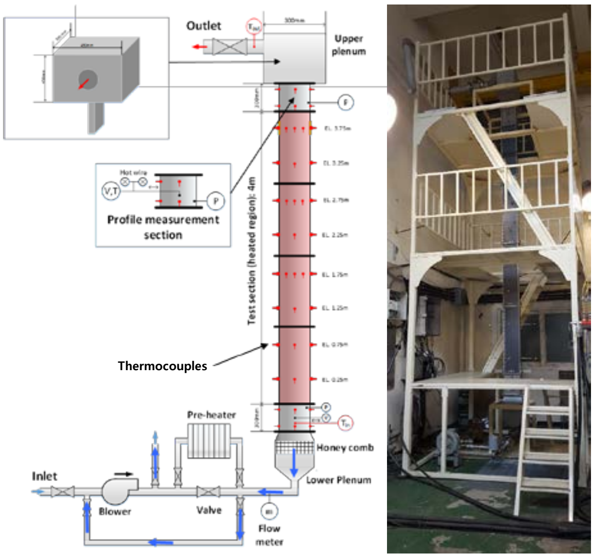

Fig. 3. The schematic diagram (left) and picture (right) of the SNU experimental facility

Besides an experiment using gas PIV method is being prepared with the test section of 1m length. The purpose of this experiment is to identify velocity profile that affects heat transfer and to validate the CFD calculation result. Fig. 4 shows the design of natural circulation visualization test facility and its experiment. Four walls of the test section will consist of four sheets of heat resistant glass. For the heat flux from glass wall, transparent conducting material, FTO (Fluorine doped Tin Oxide), will be coated to the one side of glass and used as a heat source with power. DEHS (Di-Ethyl-Hexyl-Sebacat) aerosol which is non-toxic and volatile oil will be injected to the test section as seed particles for PIV method.

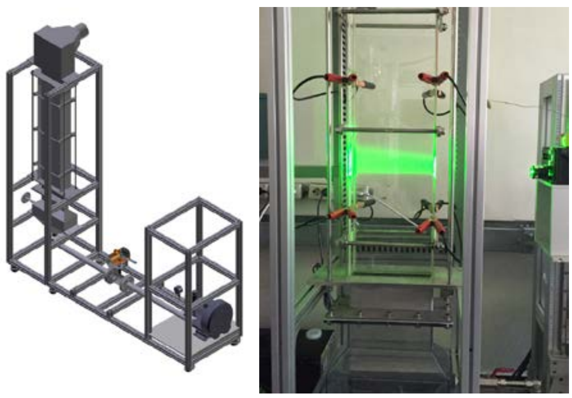

Fig. 4. The design of natural circulation visualization test facility (left) and its experiment (right)

3.2 CFD calculation with RANS models

As CFD calculation compensates the experimental result and gives help to comprehension of the phenomena, it is necessary. As mentioned above, however, the heat transfer regime corresponding to the flow inside the RCCS riser is guessed to be a mixed convection turbulent flow. The Reynolds-Averaged Navier-Stokes (RANS) turbulence models have difficulty to calculate the ascending mixed convection flows due to deformed velocity profiles near the wall and careful modeling and its validation is required. [11]

Therefore, CFD benchmark calculation for strongly heated gas flow will be performed to adopt the optimum turbulence model among the Reynolds-Averaged Navier-Stokes (RANS) models. Benchmark calculation will be performed with commercial CFD code, STAR-CCM+ 10, which has 12 turbulence RANS models except for Reynolds stress turbulence models. There are 8 k-epsilon models, 2 k-omega models and 2 Spalart-Allmaras models.

Benchmark calculation will be performed with Shehata's experimental data which has boundary conditions similar to the present study's experimental conditions. He performed experiments for air flowing upward in a vertical tube that heating rates cause significant property variation in primarily forced convection. [12] Conditions of the benchmark calculation will be three dimensional, steady state and variable properties flow. Mesh validation will be performed with Richardson extrapolation and y+ values of near wall cells. Wall temperature distributions of test section and velocity and temperature structures of the internal flow will be compared with the CFD calculation results. Based on this benchmark calculation, the CFD calculation for the RCCS riser will be conducted.

4. Conclusions

In this study, preliminary analysis on the convective phenomena inside the RCCS riser and its experimental facilities were conducted. Based on the scaling law, the thermal and hydraulic conditions for each experimental facility were derived and its heat transfer regimes were identified as mixed convection. Also, through CFD calculation, it was found that the heat flux distribution inside the riser is dissimilar to that on the outside of the riser.

Based on these facts, the experiment and CFD analysis for the convection phenomena inside the RCCS riser are being prepared and its result will be utilized to verify the performance of RCCS.

Acknowledgement

This research was supported by the National Nuclear R&D Program through the National Research Foundation of Korea(NRF) funded by MSIP; Ministry of Science ICT & Future Planning
(No. NRF-2015M2A8A2076525)

References

[1] J. H. Chang, Y. W. Kim, K. Y. Lee et al., A study of a nuclear hydrogen production demonstration plant, Nucl. Eng. Technol., Vol. 39, p. 111, 2007.

[2] J. H. Kim, Y. Y. Bae, S.D. Hong., et al., Results of the Preliminary Test in the 1/4-Scale RCCS of the PMR200 VHTR, Trans. of the KNS autumn meeting, Oct.30-31, 2014, Pyeongchang, Korea.

[3] S. Lomperski, W. D. Pointer, C. P. Tzanos, et al., Generation IV Nuclear Energy System Initiative. Air-Cooled Option RCCS Studies and NSTF Preparation (ANL-GenIV-179), Argonne National Laboratory, Argonne, IL 2011.

[4] D. Lisowski, M. Moses, M. H. Anderson, M. L. Corradini, Design Considerations for a Scaled RCCS with Air for the VHTR, Proceedings of NURETH 15, May 12-15, 2013, Pisa, Italy.

[5] K. L. McVaya, J. H. Park, S. Lee, et al., Preliminary tests of particle image velocimetry for the upper plenum of a scaled model of a very high temperature gas cooled reactor, Progress in Nuclear Energy, Vol. 83, p. 305, 2015

[6] C. K. Jo, H. S. Lim, J. M. Noh, Preconceptual Designs of the 200 MWth Prism and Pebble-bed Type VHTR Cores, Proc. Int. Conf. Physics Reactors, September 14-19, 2008, Interlaken, Switzerland

[7] Y. Y. Bae, B. H. Cho, Scaling Analysis of PMR200 RCCS Natural Cooling Performance Test (KAERI/GP-415/2015), Korea Atomic Energy Research Institute, Daejeon, 2015

[8] D. D. Lisowski, M. T. Farmer, S. Lomperski, et al., Design Report for the ½ Scale Air-Cooled RCCS Tests in the Natural convection Shutdown heat removal Test Facility (NSTF) (ANL-SMR-8), Argonne National Laboratory, Argonne, IL 2014.

[9] J. I. Lee, P. Hejzlar, P. Saha, et al., Studies of the deteriorated turbulent heat transfer regime for the gas-cooled fast reactor decay heat removal system, Nucl. Eng. Des., Vol. 237, p. 1033, 2007

[10] B. Metais and E. R. G. Eckert, Forced, Mixed and Free Convection Regimes, ASME J. Heat Transfer, Vol. 86, p. 295, 1964

[11] A. Keshmiri, M. A. Cotton, Y. Addad et al., Turbulence Models and Large Eddy Simulations Applied to Ascending Mixed Convection Flows, Flow Turbulence Combust, Vol. 89, p. 407, 2012

[12] A. M. Shehata, Mean Turbulence Structure in Strongly Heated Air Flows, Ph.D. dissertation, University of Arizona, Tucson, AZ, 1984.