Assessment of Scalability from Numerical Simulation of Air Flow through RCCS Riser of NACEF and NSTF

Yoon-Yeong Bae[a*], Chan-Soo Kim[a], Jong-Hwan Kim[a], Eung-Seon Kim[a], Minhwan Kim[a]
[a]*Korea Atomic Energy Research Institute, 111 Daedeok-daero 989, Yuseong, Daejeon, Korea 34057*
[*]*Corresponding author: yybae@kaeri.re.kr*

1. Introduction

The reactor cavity cooling system (RCCS) is one of the key design features of gas-cooled reactor such as PMR200 and MHTGR, which passively removes heat from reactor in order to maintain the temperature of the reactor surface and reactor cavity concrete wall below the design limits.

Under the joint research program between KAERI and ANL, the scalability from a model to a prototype was assessed based on the scaling analysis performed earlier at KAERI [1]. Numerical simulations were performed by using the experimental data obtained from the KAERI's quarter-scale test facility, NACEF, and ANL's half-scale test facility, NSTF, (both are scaled down only in the vertical direction) in order to provide the detailed information required for the assessment of scalability.

Scalability between models and prototype was assessed by comparing the results of numerical simulation of air flow through a single riser of NACEF and NSTF. The heat transfer modes of air flows in the riser were mostly placed in the mixed convection regime; and it made a simple inter or extrapolation difficult.

This paper describes the results of numerical simulations and addresses the scalability identified from the analysis of fluid-thermal behavior in the riser.

2. Numerical Simulation

In this section, after identifying the problem to be solved, the governing equations are described, followed by discussion of the turbulence models and the computational method.

2.1 Problem Identification

The computational domain is schematically shown in Figure 1, and the inlet flow conditions (velocity, temperature and Reynolds number) are summarized in Table 1.

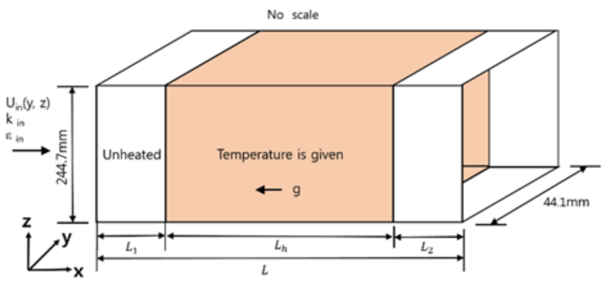


Figure 1. Schematic computational domain.

The vertical upward flow direction is aligned with the x-coordinate. The cross-sectional dimensions are the same for two test facilities, and only the total (unheated and heated) length is different. The temporally-constant and spatially-varying temperature conditions, which were obtained from the experiments, were applied to the riser sides ($z = 0$ and $z = z_L$). At the side walls ($y = 0$ and $y = y_L$), the average temperatures of the two sides were applied. The symbol y_L and z_L represent the end positions in the y and z directions. Although there are unheated sections at the inlet and outlet, the temperature boundary conditions were imposed over the whole length.

Table 1. Summary of inlet and boundary conditions.

	KAERI		ANL	
Case	$Pl_R = 1$	$Ri_R = 1$	$Pl_R = 1$	$Ri_R = 1$
ID	NACEF-4	NACEF-5	NSTF-2	NSTF-1
u_{in}, *m/s*	0.98	1.8	1.46	2.35
T_{in}, *K*	290	290	290.8	295.5
Re_{in}	4455	8167	7219	11210
L_1, *m*	1.0		0.17	
L_h, *m*	4.0		6.83	
L_2, *m*	0.0		0.40	

2.2 Governing Equations

The governing equations are the three-dimensional Reynolds-averaged Navier-Stokes equations and energy conservation equation in Cartesian coordinates.

2.3 Turbulence Models

The turbulence models used in this study were selected to obtain the best performance in each cases, which were RNG k-ε model (RNG), low-Re k-ε model and Realizable k-ε model. The low-Re k-ε model (MK) is the one proposed in [2]. Except for MK, all turbulence models adopted the two-layer model, where the wall layer is solved with one-equation model.

Table 2. Turbulence models performed best in each case.

	KAERI		ANL	
Case	$Pl_R = 1$	$Ri_R = 1$	$Pl_R = 1$	$Ri_R = 1$
Turbulence model	RNG-TL	MK	RKE-TL	RKE-TL

2.4 Computational Method

The numerical simulations were conducted by using an in-house code, which showed a good performance in the authors' earlier work [3].

The governing equations were discretized by the finite volume method (FVM) and solved by the SIMPLE algorithm with a single pressure correction step.

The grid numbers in x, y and z directions were 200, 40 and 100 respectively for the KAERI tests; and 270, 40 and 100 respectively for the ANL tests.

The value of y^+ at the first node from the wall, y_P^+ , was retained to be smaller than 0.5, which has been found to result in converged solutions with reasonable accuracy.

The value of k and ε at the inlet were set as $1.5u_{in}^2Ti^2$ and 1000 time of it, respectively. Ti is the turbulence intensity and u_{in} the inlet velocity. In NACEF-4 and 5, the inlet velocity profiles were obtained from a calculation repeatedly applying the exit values at the inlet. In NSTF-4 and 5, the inlet velocity profile were obtained from a laminar fully developed velocity profile. The detailed information can be found in [1] and [4].

3. Results and Discussions

In this section, after presenting the results of numerical simulation, the thermal characteristics are analyzed.

The riser is heated by radiation and convection from the reactor surface, which is simulated by a elefctrically-heated flat surface. The riser front side faces the heating panel and attains the maximum wall temperature among the sides.

3.1 Numerical Simulation of KAERI Tests

The results of numerical simulation of air flow in the NACEF riser are shown in Figure 2. The calculated bulk and centerline temperatures at the outlet (dash-dot and doted lines) agree quite well with the experimental data (solid triangle and circle).

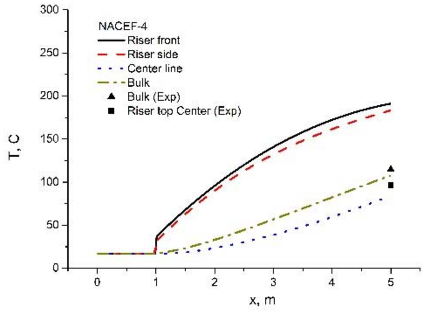
The symbols, Ri and Pl represent the Richardson and Planck number, respectively. The subscript R represents the ratio between model and prototype.

The temperatures in the case of $Ri_R = 1$ are higher than those in the case of $Pl_R = 1$. The reason for the higher temperature in the case of $Ri_R = 1$ is that the heat transfer mode in this case is more closer to mixed convection region than the case of $Pl_R = 1$.

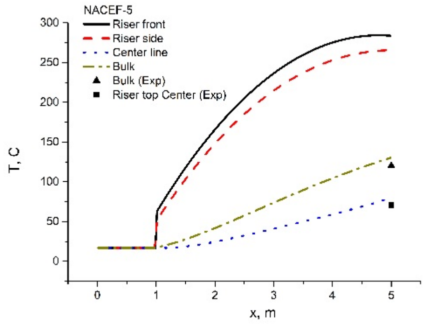
The solid square symbols in Figure 2 represent the experimental data from the NACEF; and it is very close to the computed value, indicating that the computation reasonably reproduces the experiment.

3.2 Numerical Simulation of ANL Tests

The results of numerical simulation of the air flow in the NSTF riser are presented in Figure 3. The smooth temperature rise at the front end is due to the imposed

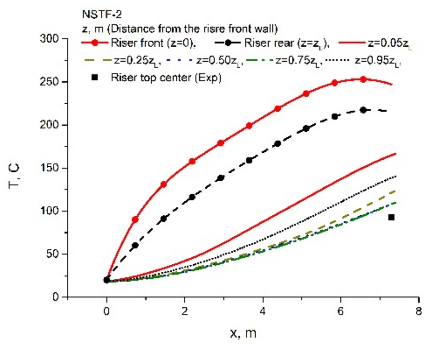


(a) $Pl_R = 1$, NACEF-4

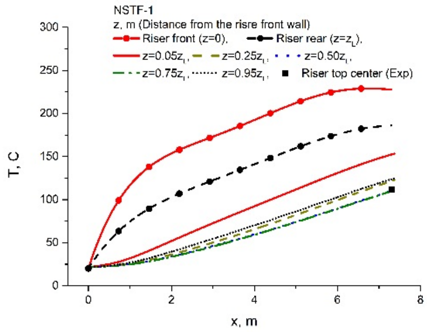


(b) $Ri_R = 1$, NACEF-5

Figure 2. Distribution of bulk and center line temperature obtained from numerical simulations.



(a) $Pl_R = 1$, NSTF-2



(b) $Ri_R = 1$, NSTF-1

Figure 3. Distribution of bulk and center line temperature obtained from numerical simulations.

temperature boundary condition in such a manner. Contrary to the simulation of KAERI test, where the case with $Ri_R = 1$ resulted in a higher temperature distribution, in the simulations of the ANL test, the case with $Pl_R = 1$ resulted in a higher temperature distribution. This unusual behavior can be explained by the analysis of heat transfer modes involved in the present work. It is the subject of the next sub-section.

The solid square symbols in Figure 3 represent the experimental data from NSTF; and it is very close to the computed value. The computation of the case of $Ri_R = 1$ exceptionally agrees well with the experimental data.

3.3 Assessment of Scalability

In Figure 4, the reduced Nusselts numbers normalized by the Nusselt number for forced convection are plotted against the buoyancy parameter, Bo, which is defined as $Bo = Gr_b/(Re_b^{2.625}Pr_b^{0.4})$ in [5]. The dimensionless numbers $Gr = g\beta(T_w - T_b)D^3/\nu^2$, $Re = vd/\nu$ and $Pr = \mu C_p/k$ are Grashof, Reynolds and Prandtl number, respectively. The subscript 'b' implies that the values are determined at the bulk conditions. The Nusselt numbers were averaged over the perimeter at a given height such as

$$\bar{h}_{cal} = \frac{\sum_{i=1}^{4}\left(q''_{w,cal,i}dA_i\right)}{\sum_{i=1}^{4}\left[\left(T_{w,i} - T_b\right)dA_i\right]} \tag{1}$$

where $q''_{w,cal,i}$ was determined from the Fourier's law at the wall. In each plot, the inlet corresponds to the upper left end-point, and as the flow proceeds upward direction the point moves right direction, returns to the left direction and then finally reach the lower left end-point.

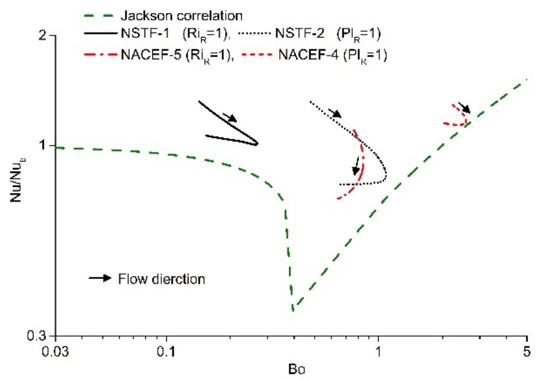


Figure 4. Variation of average reduced Nusselt number against the buoyancy number proposed by Jackson [5].

The heat transfer mode in the region around $Bo = 0.4$ is mixed convection, which is characterized by a substantial reduction in heat transfer coefficient, while the heat transfer modes in the regions left and right hand sides of the mixed convection region are forced and natural convection, respectively.

In each curve of experiment shown in Figure 4, the riser inlet corresponds to the upper left point and the outlet the lower left point. In the riser, the value of Bo increases first, then decreases, as air flows downstream.

At the beginning of the present program, it was expected that the quarter- and half-scale models, the NACEF and NSTF, of the respective prototypes will provide a way to extrapolate the data from a model test to obtain those for a prototype. However, as the experiment proceeded, we found an unusual behavior such that the Nusselt number did not show any monotonic trend, making a simple extrapolation inappropriate. It should be noted that the scaling parameter ratios $Ri_R = 1$ and $Pl_R = 1$ represent the two cases, similarity between model and prototype in riser air flow and in heat transfer in the cavity.

As shown in Figure 4, the reduced Nusselt numbers from NACEF test, designated by red dash dot and short dash, are positioned in the region of natural convection (right hand side of the lowest point of Nu/Nu_b). It is naturally expected that as the scale of model progressively increases from a quarter-scale to a half-scale, the reduced Nusselt number will decrease, and inevitably passes the region of the strongest mixed convection.

Since the actual heat transfer mode in the NSTF tests is very likely to be positioned between the conditions for $Ri_R = 1$ and $Pl_R = 1$, a mixed convection seems to have been occurred during the NSTF test. On the contrary, the heat transfer mode in the NACEF tests would largely be a natural convection.

It can also be observed in Figure 4 that the NSTF test results positioned at the locations shifted to the left direction, decreasing *Bo*, from those of the NACEF tests. From this fact it can be conjectured that the values of *Bo* for the prototype should be smaller than those for NSTF. Table 3 summarizes the values of Nu/Nu_b and *Bo* obtained from the numerical simulations.

Table 3. Numerical values of Nu/Nu_b and *Bo* at the riser exit.

	ANL		KAERI	
Case	NSTF-1 $Ri_R = 1$	NSTF-2 $Pl_R = 1$	NACEF-5 $Ri_R = 1$	NACEF-4 $Pl_R = 1$
Nu/Nu_b	1.064	0.781	0.714	1.158
Bo	0.154	0.635	0.643	1.939

For the sake of convenience in scaling, the buoyancy number, $Bo_q = Gr_q / \left(\mathrm{Re}_b^3 \, \mathrm{Pr}_b^{0.5}\right)$, proposed by Symolon [6], is used in the following discussions. The Grashof number is defined with heat flux instead of temperature difference such as $Gr_q = g\beta q''_w D^4 / \left(k\nu^2\right)$. Then, from the scaling analysis [7] the ratio of Bo_q between model and prototype becomes

$$\left(Bo_q\right)_R = \frac{\left(Bo_q\right)_m}{\left(Bo_q\right)_p} = \begin{cases} \ell_R^{-2} \; for \; Ri_R = 1 \\ \ell_R^{-3} \; for \; Pl_R = 1 \end{cases} \quad (2)$$

In the derivation of Eq. (2) all physical properties were considered to be constants. Once $\left(Bo_q\right)_R$ is known from Eq. (2), $\left(Bo_q\right)_R$ is computed from the experimental value of $\left(Bo_q\right)_m$. Then Nusselt number or heat transfer coefficient is estimated from the following correlation for mixed convection proposed by Seoul National University [8].

$$\frac{Nu}{Nu_T} = \left\{ \left[\frac{\left(\frac{9.2320 \times 10^{-5}}{Bo_q}\right)^{4.0330}}{1 + \left(\frac{9.2320 \times 10^{-5}}{Bo_q}\right)^{4.0330}} \right]^{4.7420} + \left(0.4756 Bo_q^{0.3326}\right)^{4.7420} \right\}^{0.2109} \quad (3)$$

where Nu_T is the Gnielinski correlation. It should be noted that Eq. (3) was developed using the test facility, which has exactly the same geometry as the RCCS riser,

Actual scaling ratio between the NSTF and NACEF is 1.7, and the expected value of $\left(Bo_q\right)_R$ between them is $1.7^2 = 2.89$. Obviously, this number is different from the value $0.643/0.154 = 4.18$ computed from the values in Table 3. The discrepancy might have arisen from distortion in scaling or inadequacy in experiment.

It is recommended that an extrapolation from model to prototype should be made with the consideration of mixed convection, when heat transfer of interest is influenced by buoyancy.

4. Conclusions

Numerical simulations were conducted for air flow through a riser of RCCS models scaled down by a half and a quarter of prototype.

A mixed convection heat transfer mode was clearly identified in the numerical simulations of both test facilities.

The heat transfer modes for both cases of $Ri_R = 1$ and $Pl_R = 1$ in the NACEF were identified as natural convection, while those in the NSTF were mixed convection for $Pl_R = 1$ and forced convection for $Ri_R = 1$.

The heat transfer mode in the prototype is expected to be forced convection slightly influenced by buoyancy, that is, weak mixed convection. However, the possibility of strong mixed convection still remains.

It is recommended that whenever heat transfer coefficients are extrapolated from experimental data strongly influenced by buoyancy to those for prototype, the possibility of appearance of mixed convection should be checked in order to avoid any false extrapolation due to non-linear behavior associated with mixed convection.

ACKNOWLEDGEMENT

The authors acknowledge the financial support from the Nuclear Research & Development Program under a National Research Foundation of Korea (NRF) grant funded by the Korean government (MSIP). (Grant codes: NRF-2017M2A8A1014758 and NRF-2014M2A8A2074090).

REFERENCES

[1] Y. Y. Bae, "Numerical simulation of air flow in RCCS riser, NHDD_KT_CA_16_004, Rev. 1," KAERI, Daejeon, 2016.

[2] H. Myong and N. Kasagi, "A new approach to the improvement of k–ε turbulence model for wall bounded shear flows," *JSME International Journal*, vol. 33, p. 63–72, 1990.

[3] Y. Y. Bae, E. S. Kim and M. H. Kim, "Numerical simulation of supercritical pressure fluids with property-dependent turbulent Prandtl number and variable damping function," *Int. J. Heat and Mass Transfer*, vol. 101, pp. 488-501, 2016.

[4] Y. Y. Bae, Numerical simulation of air flow in RCCS riser, -ANL NSTF- (unpublished)," 2017.

[5] J. D. Jackson, "Ideas for an Improved Semi-Empirical model of Turbulent Mixed Convection Heat Transfer to Fluids at Supercritical pressure," Pisa, Italy, May 12-17, 2013.

[6] P. Symolon, W. Neuhaus and R. Odell, "Mixed convection heat transsfer in smooth and vertical tubes," *Transaction of Americal Nuclear Society*, vol. 92, pp. 387-390, 2005.

[7] Y. Y. Bae, "Scaling analysis of PMR200 RCCS natural cooling performance test, NHDD-KT-CA-12-010, Rev. 2," KAERI, Daejeon, 2014.

[8] D. H. Shin, S. Y. Kim, H. K. Cho and G. C. Park, "Private communication".