Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 23-25, 2013

Numerical Simulation for Validating Scaling Laws of RCCS Model Test

Bong Hyun Cho*, Yoon Yeong Bae, Jae Man Noh
Korea Atomic Energy Research Institute, DAEDEOK-DAERO 989-111, YUSEONG-GU, DAEJEON, KOREA
**Corresponding author: bhcho1@kaeri.re.kr*

1. Introduction

The Reactor Cavity Cooling System (RCCS) is the ultimate heat sink of the core decay heat under accident conditions in HTGR. The RCCS should have a thermal capability to insure that the temperatures of internal components do not exceed the allowable maximum temperature. For a verification of the safe function of the RCCS, the heat removal capacity through the riser from the reactor cavity must be experimentally evaluated to see whether the wall temperatures of the reactor vessel surface and reactor cavity wall exceed the prescribed limits. A 1/4 height down-scale experiment on the PMR200 RCCS [1] performance is being carried out at KAERI to verify whether the RCCS has a sufficient heat removal capability required to remove residual heat in the case of a total loss of active heat removal system.

In the cavity, most of heat is exchanged by the radiation rather than convection, therefore the Planck number, which is the ration between conduction and radiation, can be a controlling non-dimensional parameter. The only way to satisfy the similarity in the Planck number is to enforce that the all ratio of the temperature rise, reference temperature, and heat flux in the model to those in the prototype are unity, that is, $\Delta T_R = 1$, $T_R = 1$, and $q''_{wR} = 1$ [1].

But in the riser, as far as the flow velocity is concerned, the dominant non-dimensional parameter is the Richardson number, the scaling of the fluid velocity is $u_R = \ell_R^{1/2}$. Radiation in the riser still dominates the heat transfer in the riser the scaling of the heat flux is also $q''_{wR} = 1$ [1].

In order to check the validity of the scaling of test section, a numerical sensitivity study was carried out using a CFD code, Fluent.

2. Methods and Results

2.1 Computational Domain

Figure 1 shows a schematic of the arrangement in the reactor cavity. The 6 risers in the cavity were simplified as a rectangular channel attached to the cavity.

The height of the prototype was set as 16 m. Reducing the vertical dimension by the 1/4 scale and ensuring space for the 6 risers, the dimension of the experimental cavity is determined as 1.4 m (W) x 4 m (H) x 0.6 m (D). H, W, and D, are the height, width, and depth, respectively. The box width, which is equivalent to the distance between the reactor vessel surface and cavity wall, was retained. The amount of heat to be removed in the case of an accident is 800 kW. The amount of heat to be removed through the six risers is 21.8 kW. Since the heated area is 0.6 m × 16 m = 9.6 m^2, the heat flux is 2.27 kW/m^2 [2].

The two-dimensional calculation domain for Fluent is constructed with hexagonal meshes less than 100,000 nodes. The top and the bottom surfaces of the cavity, and the right hand side of the riser (to be understood hereinafter as including duct) were treated as adiabatic; the left hand side of the cavity was set as a wall with constant heat flux.

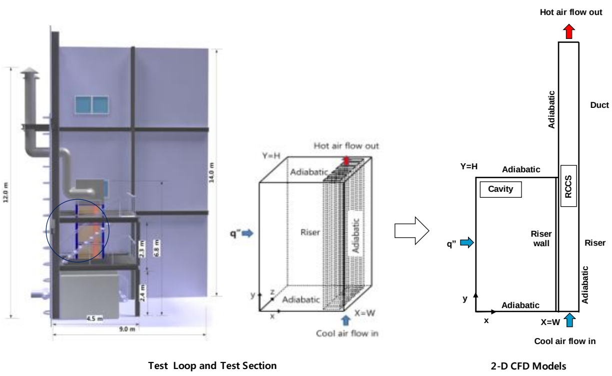

Fig. 1 Calculation domain for Fluent

2.2 Modeling for Computation

The DO radiation model embedded in Fluent is used for the calculation of radiation. The DO model requires selecting the numbers of divisions and pixels both for theta and phi. To increase the accuracy of the simulation, we chose 8 for the division and 5 for the pixel. The RNG k-ε turbulence model was used and the enhanced wall treatment was provided for the wall boundary [3]. The inlet/outlet of the riser were set as a pressure boundary, and the inlet pressure and temperature were 1 bar and 45℃, respectively. The NIST data base was used for air property and the variation of properties due to the temperature change was considered.

2.3 Calculation Results

Fig. 2 shows the performance of prototype RCCS which has a heated section with a height of 16m. The temperature of reactor vessel wall increased steadily from the bottom to the top and reached an average temperature about 270-280℃. The reactor vessel wall temperature has a sharp peak at the cavity top. The temperature along the riser wall reaches about 100℃, and then the radiation heat transfer effect fairly diminished. The velocity in the riser center plane was 5.8m/s at the inlet, and increased to 6.8 m/s at the outlet . The velocity became stable when the fluid reached to the unheated section of the riser.

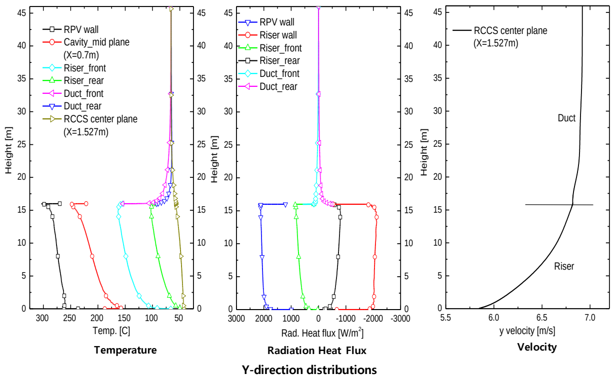

Fig. 2 Performance of the prototype RCCS

Fig. 3 shows the results of calculations for checking a sensitivity to heat flux in a model test section. The wall temperature distributions along the cavity wall in the model with the same heat flux as that of the prototype (Model_1q) shows a good agreement with those in the prototype. It may imply that radiation heat transfer dominates in the cavity. However there are different results for the fluid temperature and velocity in the riser. The temperature distribution of model case of four times the heat flux for the prototype (Model_4q) has similar trend with that of prototype at the riser center plane; while the velocities of all cases are fairly lower than those in the prototype. The velocities in the Model_1q are about half of those in the prototype.

Fig. 4 shows the sensitivity of the calculations to heated height. The results shows a similar trend to that to heat flux.

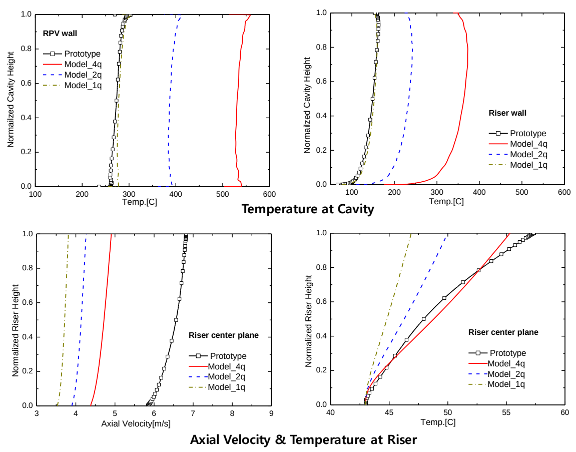

Fig. 3 Sensitivity to heat flux

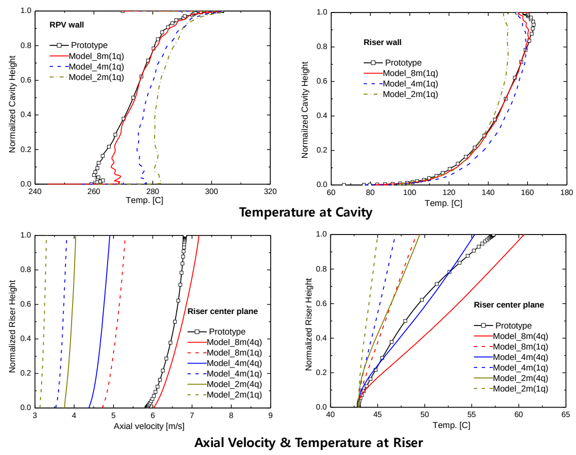

Fig. 4 Sensitivity to heated length

3. Conclusions

Numerical simulations were performed to check the validity of the scaling laws of the test section for the PMR200 RCCS performance test. Sensitivity to the three variables, such as heat flux, heated length and riser width, were studied. The calculated temperature distributions along the bilateral walls of the reactor cavity and riser showed an exceptionally good agreement between the model case, Model_1q, and prototype. This implies that the scaling law, $q''_{wR} = 1$, in the cavity and riser are almost correct.

Numerical calculation confirmed that the velocity scaling derived from the similarity in the Richardson number, $u_R = \ell_R^{1/2}$ holds when $q''_{wR} = 1$.

The scaling laws and results of the numerical calculations will be compared validated with the experiment to be carried out in the next year.

Acknowledgements

This work was supported by Nuclear R&D Program of the National Research Foundation of Korea (NRF) grant funded by the Korean government (MEST).

REFERENCES

[1] Bae, Y. Y., “Scaling analysis of PMR200 RCCS natural cooling performance test,” Calculation Note, NHDD-KT-CA-12-010, Rev. 1, 2013.

[2] Bae, Y.Y. Cho, B.H., Hong, S.D., Kim, Y. W. “Numerical simulation of Convective and Radiative Heat Transfer in the PMR200 Reactor Cavity,” Proceedings of the 15th Int. Topical Meeting on Nuclear Reactor Thermal Hydraulics, NURETH-15-091, Pisa, Italy, May. 12-17, 2013.

[3] FLUENT User’s Guide, Fluent Inc., 2006