Development of a Preliminary Numerical Analysis Methodology for Predicting Natural Circulation Cooling Performance Based on NSTF Experiments

Nahid Hasan [a*], Jophous Mugabi [a], Jae-Ho Jeong [b], Sin-Yeob Kim [c]
[a]*Department of Mechanical Engineering, Gachon Univ., 1332, Seongnam-daero, Sujeong-gu, Seongnam-si, Gyeonggi-do 13306, Republic of Korea*
[b]*Department of Mechanical Engineering, Chung-Ang University., Dongjak-gu, Seoul, 06974*
[c]*Korea Atomic Energy Research Institute., Yuseong-gu, Daejeon, 34057*

[*]*Corresponding author: jaehojeong@cau.ac.kr*

***Keywords :** HTGR, RCCS, CFD, Preliminary analysis*

1. Introduction

The High-Temperature Gas-cooled Reactor (HTGR) is an advanced reactor technology designed to operate stably at high temperatures, making it suitable for efficient power generation and industrial heat supply. One of the key passive safety systems in HTGR is the Reactor Cavity Cooling System (RCCS), which removes residual heat from the reactor core without the need for external power or additional coolant during emergency conditions.

RCCS passively removes approximately 0.3–0.6% of the reactor's rated power in the event of an accident, ensuring that the temperature of the reactor containment concrete does not exceed design limits even under normal operating conditions. The primary design objective of the RCCS is to maintain an appropriate temperature for both the reactor pressure vessel and the concrete structures inside the containment under normal and accident conditions [1].

In particular, the Argonne National Laboratory (ANL) in the United States conducted a half-scale height experiment to validate the RCCS performance in the Modular High-Temperature Gas-cooled Reactor (MHTGR). Based on this experimental data, numerical methodologies have been developed to predict natural convection cooling performance.

In this study, preliminary analyses were conducted to establish boundary conditions and radiation heat transfer models for simulating natural circulation within the RCCS.

2. Numerical Methodology

The CFD model has been developed for a 1/2 scale model of the MHTGR using the commercial CFD software ANSYS CFX. The numerical simulation of the 1/2 scale M-HTGR NSTF experimental facility is described. The results are compared with and analyzed against the experimental data from the NSTF facility.

2.1 Analysis Model

A schematic diagram of the NSTF configuration is shown in Figure 1. The NSTF experimental facility is a 1/2-scale model of the RCCS in the MHTGR reactor, designed to evaluate the cooling performance of the RCCS under various operating conditions [2].

The operating principle of the NSTF is based on natural circulation. Ambient air enters through the inlet downcomer and flows upward through the riser duct, where it absorbs heat through convective and radiative heat transfer within the cavity. Due to the temperature difference, a density gradient is generated, driving the heated air to exit through the outlet chimney, thereby maintaining continuous natural circulation.

The NSTF is an air-cooled cooling system that passively removes residual heat from the reactor using natural air circulation. Radiative heat transfer dominates in the cavity due to the high temperatures emitted by the reactor pressure vessel [3]. The heat transfer mechanism inside the cavity and the natural circulation flow path are illustrated in Figure 2.

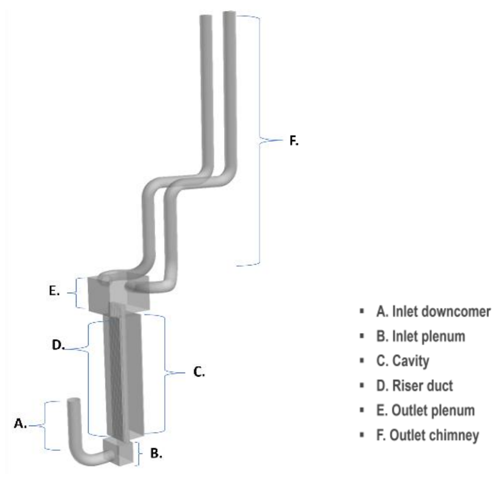

Fig. 1. Schematic of overall NSTF

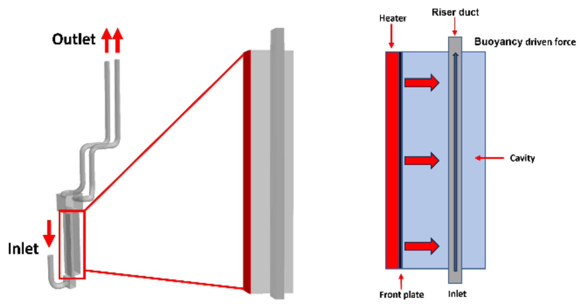

Fig. 2. Heat Transfer and Air Flow Path in the NSTF

2.2 Governing Equations for Radiative Heat Transfer

The Radiative Transfer Equation (RTE) is presented in Equation (1). The RTE describes the variation of radiation intensity within a medium due to absorption, emission, scattering, and external radiation sources. Radiation intensity decreases due to absorption (K_a) and scattering (K_s) within the medium, while emitted radiation follows Planck's blackbody radiation law. Additionally, the phase function (Φ) governs the redistribution of scattered radiation in different directions, while external sources such as flames or radiation emissions may further influence the radiation intensity. CFX applies various radiation models based on this equation to analyze radiative heat transfer.

Table 1 presents the characteristics of radiation models available in CFX and their suitability for RCCS analysis. Based on this, we assumed that the cavity is filled with transparent air and conducted the radiation heat transfer analysis using the Surface-to-Surface (S2S) model.

Table 1: Comparison of Radiation Models

Radiation Model	Computation cost	Accuracy	RCCS Appropriateness
P1	Low	Moderate	2/5
Rosseland	Moderate	Low	1/5
DTM	High	High	4/5
Monte Carlo	Very High	Very high	5/5

The S2S model neglects absorption and scattering within the medium and only considers surface-to-surface radiation exchange. This model relies on Equation (2) and the calculation of view factors to determine radiative heat transfer between surfaces.

$$\frac{dI_v(r,s)}{ds} = (-(K_{av} + K_{av})I_v(r,s) + K_{av}I_b(v,T) + \frac{K_{sv}}{4\pi}\int dI_v(r,s')\Phi(s \cdot s')\,d\Omega' + S) \quad (1)$$

$$Q_{k,in} = \sum_{j=1}^{N} F_{jk}\,Q_{j,out} \quad (2)$$

2.3 Boundary conditions

The boundary conditions for the NSTF model are presented in Table 2. The Monte Carlo radiation model was employed, specifically utilizing the Surface-to-Surface (S2S) mode.

At the inlet and outlet, opening and pressure boundary conditions were applied to facilitate natural circulation driven by buoyancy effects. These conditions ensure that airflow is passively induced and expelled within the system, allowing for a more accurate simulation of the RCCS cooling mechanism.

Additionally, in this study, the commercially available k-ε turbulence model was applied. To account for buoyancy effects, the Buoyancy Production term was activated, enabling a more precise representation of the impact of temperature-induced buoyancy forces on turbulent kinetic energy.

Furthermore, a structured grid with 3.3 million cells was generated and utilized in the simulation. This high-resolution grid system enhances the accuracy of the computational domain and ensures reliable simulation results.

Table 2: Boundary Condition

Parameter	Value [Unit]
y+	30< y+ <100
Turbulence Model	k-ε
Radiation Model	Monte Carlo
Radiation Mode	Surface to Surface
Air Inlet [Opening, Static Pressure]	1 [atm]
Air Outlet [Opening, Static Pressure]	1 [atm]
Heater Heat flux	6088 [W/m^2]

3. Results

The velocity field inside the cavity was illustrated in Figure 3 to investigate the dominant role of radiative heat transfer within the cavity.

A comparison between the simulations with and without the activation of the radiation model revealed that the average internal velocity was 0.639 m/s without radiation and 0.246 m/s with radiation.

Furthermore, as shown in Table 3, a comparison of the heat flux and radiative flux inside the cavity indicated that when the radiation model was activated, radiative heat transfer accounted for 82% of the total heat transfer.

This result is attributed to the energy balance, where the inclusion of radiative heat transfer led to a reduction in the contributions of natural convection and conduction to the overall heat transfer.

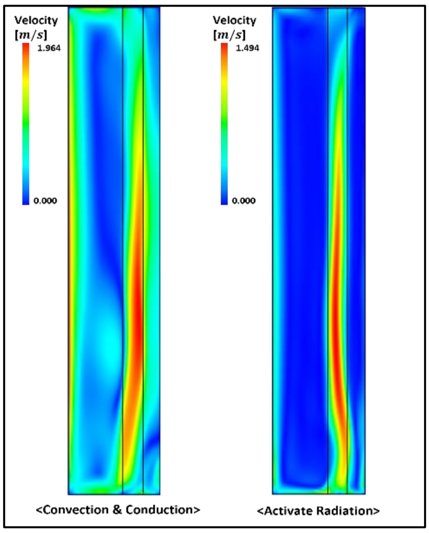

Fig. 3. Cross-sectional velocity distribution of the cavity

Table 3: Analysis of Heat Transfer With and Without Radiation

	Convection& Conduction	Activate Radiation
Velocity in cavity	0.639 [m/s]	0.246 [m/s]
Wall heat flux	6078 [w/m^2]	5990 [w/m^2]
Radiative flux	0	5219 [w/m^2]

The CFD results were further compared with the experimental data, as presented in Table 4. The outlet temperature, mass flow rate, and front plate temperature exhibited an error within 10%. Although the error rate was 30% due to a 7 Pa deviation in pressure difference, the absolute value of 7 Pa is relatively small and considered insignificant in practical applications [5].

Table 4: Comparison of Experimental and CFD Results

	Experiment Results	CFD Results	Error
Outlet Temperature	119.9 [℃]	113 [℃]	5.75%
Mass flow rate	0.548 [kg/s]	0.607 [kg/s]	9.67%
Riser duct ΔPa	22 [Pa]	29 [Pa]	31.8%
Front plate	393.9 [℃]	432.1 [℃]	9.84%

4. Conclusions

This study developed a numerical analysis methodology to predict the natural circulation cooling performance of the Reactor Cavity Cooling System (RCCS) based on NSTF experimental data. The findings validate the applicability of the proposed numerical approach for evaluating RCCS cooling performance and provide critical insights into the thermal behavior of natural circulation cooling systems under high-temperature conditions.

Based on this study, CFD analysis will be conducted on a water-cooled RCCS, which is expected to be applied to the design of next-generation High-Temperature Gas-cooled Reactors (HTGRs).

Acknowledgements

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIT) (No. RS-2024-00457356)

REFERENCES

[1] Kim, Chan Soo, Lee, Sung Nam, Kim, Jong Hwan, and Bae, Yoon Yeong, "Analytical Studies to Simulate Test Results for Reactor Cavity Cooling System at High Temperature Gas-cooled Reactor," KSME17-Th11E001, 2017.

[2] Lisowski, D. D., Farmer, M. T., Lomperski, S., Kilsdonk, D. J., Bremer, N., and Aeschlimann, R. W., "Design Report for the ½ Scale Air-Cooled RCCS Tests in the Natural Convection Shutdown Heat Removal Test Facility (NSTF)," Proceedings of the HTR 2014, 2014.

[3] Lisowski, D. D., Kraus, A. R., Bucknor, M. D., Hu, R., and Farmer, M. T., "Experimental Observations of Natural Circulation Flow in the NSTF," Argonne National Laboratory, Nuclear Engineering Division, Lemont, IL, USA.

[4] Tzanos, C. P., "CFD Analysis for the Applicability of the Convection Shutdown Heat Removal Test Facility (NSTF) for the Simulation of the VHTR RCCS," Argonne National Laboratory Document, 2005.

[5] Matthew Bucknor, Rui Hu, Darius Lisowski, Adam Kraus, "Comparisons of RELAP5-3D Analyses to Experimental Data from the Natural Convection Shutdown Heat Removal Test Facility," Proceedings of ICAPP 2016, San Francisco, CA, USA, April 17-20, 2016.