INL/CON-24-77045-Revision-0

High Fidelity Simulations of Air-Cooled Reactor Cavity Cooling System

May 2024

Changing the World's Energy Future

Sinan Okyay, Victor Coppo Leite, Paolo Balestra, Gerhard Strydom, Elia Merzari

INL/CON-24-77045-Revision-0

High Fidelity Simulations of Air-Cooled Reactor Cavity Cooling System

Sinan Okyay, Victor Coppo Leite, Paolo Balestra, Gerhard Strydom, Elia Merzari

May 2024

Idaho National Laboratory
Idaho Falls, Idaho 83415

http://www.inl.gov

Prepared for the
U.S. Department of Energy
Under DOE Idaho Operations Office
Contract DE-AC07-05ID14517

High-Fidelity Simulations of an Air-cooled Reactor Cavity Cooling System

Sinan Okyay [1,2,*], Victor Coppo Leite[1], Elia Merzari[2], Paolo Balestra[1], Gerhard Strydom[1]

[1]Idaho National Laboratory, Idaho Falls, ID
[2]The Pennsylvania State University, State College, PA

[leave space for DOI, which will be inserted by ANS]

ABSTRACT

Nuclear energy is increasingly being acknowledged as having a pivotal role in the global shift toward cleaner energy solutions. Among the various types of Generation IV reactors, advanced nuclear technologies such as high-temperature gas-cooled reactors (HTGRs) are particularly appealing, thanks to their high-temperature heat output and potential for cogeneration.

HTGR designs incorporate passive safety systems (e.g., the Reactor Cavity Cooling System [RCCS]) that utilize natural principles to manage heat dissipation from the reactor pressure vessel (RPV) during accidents or routine shutdowns. Regulatory bodies require thorough validation of such safety systems in order to ensure they meet specified standards. Consequently, the industry is experiencing a pressing need for advanced simulation tools that can accurately assess the performance of these types of systems. In the literature, a knowledge gap exists concerning high-fidelity data for the RCCS, and this gap is one of the areas of focus of the present study. This research focuses on a specific RCCS designed for General Atomics' Modular High-Temperature Gas Reactor (GA-MHTGR). Experimental studies on a scaled version of the air-cooled RCCS used in GA-MHTGR were conducted by the University of Wisconsin-Madison (UW-Madison).

This work contributes to a broader initiative aimed at establishing a numerical benchmark based on the UW-Madison experiments. As a first step, we performed high-fidelity simulations of the experimental facility setup in order to analyze the flow physics in such systems and to validate NekRS and the Multiphysics Object-Oriented Simulation Environment (MOOSE) heat transfer and radiation modules.

Keywords: air-cooled RCCS, NekRS, high-fidelity CFD

1. INTRODUCTION

Nuclear energy is increasingly recognized as a critical player in the global transition to clean energy. It can serve as a reliable, stable energy source, and may significantly impact mitigating climate change by reducing carbon emissions. Advanced nuclear technologies such as Generation IV reactors represent a distinctive solution in this area, thanks to their enhanced safety, sustainability, efficiency, and cost-effectiveness.

High-temperature gas-cooled reactors (HTGRs) are among the most appealing Generation IV reactor designs, due to their high thermal output and potential cogeneration capabilities. HTGRs have attracted renewed interest thanks to their exceptional passive safety systems and ability to provide process heat, as outlined by the U.S. Department of Energy (DOE) [1]. HTGR passive safety systems, which often rely on fundamental

*sinan.okyay@inl.gov

principles of nature, do not employ compressors, pumps, or other active components. This means minimal to no human intervention is required to operate them under accident conditions [2].

These reactor concepts rely on a Reactor Cavity Cooling System (RCCS) to carry out heat removal from the reactor pressure vessel (RPV), whether in accident scenarios or during normal shutdown. Typically, these systems are designed to operate with air or water as the working fluid (for the present study, air was selected). In brief, the RCCS includes vertical rectangular ducts called ”risers” (or riser ducts) around the RPV. These risers receive the heat from the RPV through convection and radiation heat transfer to the risers [3]. The air inside the risers transfers the heat coming from the RPV to the environment via natural convection, thus enabling the system to operate passively.

Unlike other mechanisms, radiation heat transfer manifests despite the absence of a material medium [4]. It is a complex phenomenon governed by many different design parameters (e.g., system geometry, absorptivity, and emissivity [5]). At the same time, regulators require accurate demonstrations verifying that safety systems such as the RCCS can meet their specifications. Hence, the development of tools for evaluating these effects is an actual demand within the industrial community.

In this context, several experimental and numerical studies were conducted in [2, 6, 7, 8, 9, 10] to improve the understanding of thermal hydraulics in this area. The Argonne National Laboratory (ANL) Natural Convection Shutdown Heat Removal Test Facility (NSTF) provided valuable experimental and numerical data on modeling the RCCS designed for General Atomics' Modular High-Temperature Gas Reactor (GA-MHTGR) [6, 7, 8], and computational fluid dynamics (CFD) tools were used to enhance the performance of system-level thermal-hydraulic analysis codes in [8, 9]. However, the literature reflects a gap in regard to high-fidelity data on the RCCS, and this is one of the present study's areas of focus. Note that the literature does contain high-fidelity CFD simulations for partial geometries of RCCS, including large-eddy simulations (LES) of risers for air-cooled RCCS [11].

The present study focuses on a specific air-cooled RCCS designed for GA-MHTGR. The University of Wisconsin-Madison (UW-Madison) conducted experiments on a scaled version of this RCCS [2], and the present work is part of a broader effort to establish a numerical benchmark based on the experiments conducted at UW-Madison [2]. Here, we offer validation of the fundamental aspects of the modeling strategy. More specifically, the models developed in this work validate the modeling preferences of the RCCS's main physical aspects (i.e., buoyancy effects, radiation heat transfer, and turbulence).

We performed high-fidelity simulations to analyze the flow physics in the system. To the best of our knowledge, this is something that has never been done before. Thus, this work represents an important contribution to the community, as the flow patterns and buoyancy effects dictate the flow circulation in air-cooled RCCS designs. Additionally, we present a simulation that focuses on the heat transfer in the UW-Madison RCCS experimental facility.

2. METHODS

This section outlines the methodology employed in the present study. The RCCS test facility and experiments are summarized, followed by an explanation of the modeling strategy and the numerical tools utilized. Two numerical models are presented in this study: a thermal model of the heated cavity, and a high-fidelity thermal-hydraulic model of the risers.

2.1. RCCS Test Facilty and Experiement Summary

In the present work, a scaled experiment of the GA-MHTGR RCCS was utilized as a reference facility. This experiment consisted of three main components: the inlet plenum, heated cavity, and outlet plenum/exhaust ducts (see Figure 1). The inlet plenum is the entry point for air drawn from the environment by the heated air in the RCCS. Within the heated cavity, electrical resistance heaters simulate the RPV and emit heat to the six riser ducts. Finally, the outlet plenum serves as a mixing space prior to the heated air being released back into the environment via a pair of exhaust ducts. The orange arrows in Figure 1 indicate the airflow direction inside the RCCS.

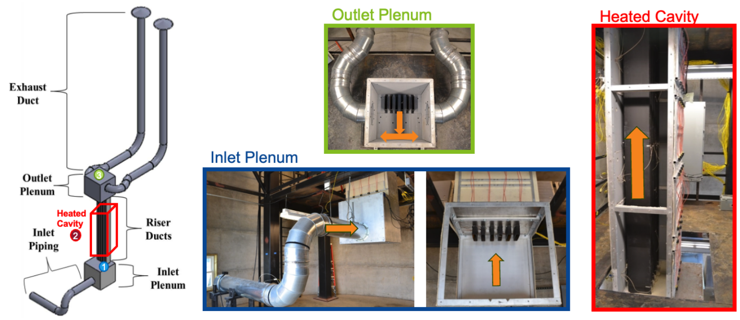

Figure 1. Facility overview (images taken from [2]).

Two types of heat transfer regimes were experimentally modeled in this facility: forced and natural convection. Natural convection tests were conducted with both uniform and asymmetric flux in order to investigate the effect of power shape, whereas the forced convection tests were conducted using a uniform flux profile only. Each type of test was run at two different power levels and repeated two times to analyze repeatability. Only one of the four tests, conducted under the uniform flux scenario at low power, showed unstable behavior (i.e., flow reversal present in the exhaust ducts). Table I summarizes the experiments. The low-power forced convection test (labeled test #14 in [2]) was selected as the starting point for modeling this facility.

Table I. Experiment summary.

Type	Case	# of Experiments	Instability
Forced Convection	Forced Flow	4 (2 Power Levels, 2 Repeat)	No
Natural Convection	Uniform Flux	4 (2 Power Levels, 2 Repeat)	No (1 exception)
	Asymmetric Flux	4 (2 Power Levels, 2 Repeat)	Yes

2.2. Numerical Tools

The Nuclear Energy Advanced Modeling and Simulation (NEAMS) program, operated under the auspices of the DOE Office of Nuclear Energy, plays a crucial role in advancing nuclear technology by developing cutting-edge modeling and simulation tools intended to facilitate the development of next-generation nuclear technology and enhance our understanding thereof. The present work will utilize two numerical tools from the NEAMS program: The Multiphysics Object-Oriented Simulation Environment (MOOSE) and NekRS.

2.2.1. MOOSE

Developed by Idaho National Laboratory, MOOSE is a framework for performing multi-scale and multi-physics simulations [12]. It can apply Newton's method to solve coupled linear and non-linear partial equations. The MOOSE framework implementation primarily relies on the finite element method, which seeks solutions to equations by employing a set of shape functions in order to obtain the solution for a system of equations [13]. In the present study, the MOOSE heat transfer module was employed to create a thermal model of the RCCS heated cavity system (see Section 2.3.1).

2.2.2. NekRS

NekRS, an open-source code for solving CFD problems [14], is employed by the NEAMS program to perform advanced nuclear reactor thermal-hydraulic analyses at various fidelity levels. NekRS was developed by ANL for simulating transitional and turbulent flows in complex geometries [14]. It employs the spectral element method, a high-order weighted residual technique that combines the geometric flexibility of finite elements with the rapid convergence and computational efficiency of global spectral methods [15]. NekRS is designed to operate based on GPUs, providing solutions to problems that demand high-computational performance. NekRS will be used to create a high-fidelity simulation of the air flow in the RCCS design.

2.3. Numerical Models

2.3.1. Heated Cavity

The heated cavity is vital to the air-cooled RCCS facility, acting as a thermal enclosure for transferring heat from the heaters to the six riser ducts and also limiting any heat losses to the environment. As with the reactor cavity of the GA-MHTGR, natural convection cells develop within the heated cavity. Heat transfer primarily occurs through radiation and convection. Conduction occurs between the heaters and the frame of the heated cavity, as the heaters are mounted within the cavity frame. The total power for the reference test that is going to modeled is 19.82 kW. Figure 2 presents a heat diagram of the heated cavity system.One should note that in the Figure 2, the heat goes to the frame is lost to the environment.

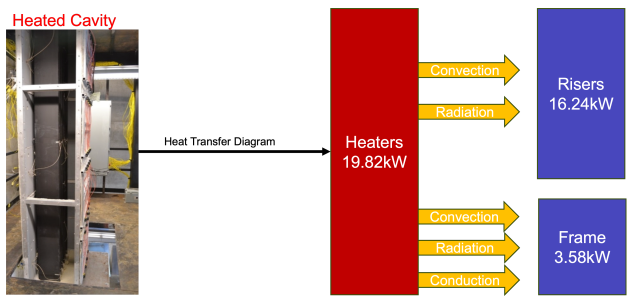

Figure 2. Heat transfer diagram of the heated cavity system.

MOOSE was used to simulate the system presented in the figure above. Radiation and conduction were modeled by leveraging the respective MOOSE modules to solve the heat balance equations.

$$\dot{q} = -\nabla \cdot (k \nabla T) \tag{1}$$

where T is temperature, k is the thermal conductivity, and $\dot{q}$ is a heat source. The total power is distributed to each heaters as a uniform volumetric heat source. Convection, on the other hand, was modeled simply as convective boundary conditions, which imply a Robin boundary condition at the specified location. Below, the heat transfer coefficients are summarized for the thermal model. As a first approximation, a correlation valid for tall cavities was utilized to calculate the heat transfer coefficient for the cavity interior [16]. Since the heated cavity is an enclosed box, the heat removed by the convective boundary condition for the inside of the cavity was reapplied to the system as a Neumann boundary condition in order to satisfy the energy balance. Radiative heat transfer was modeled using the net radiation method in the MOOSE modules. On the other hand, the view factors for the net radiation transfer method were produced using the ray tracing module. Additional information can be obtained from the following reference [12]. The material properties and geometric specification were taken from the experimental report [2].

Table II. Heat transfer coefficient for the thermal model.

Heat Transfer Coefficient Data	Heat Transfer Coefficient ($W/(m^2K)$)	Bulk Temperature (K)
Riser Ducts	25	323
Cavity Inside	6.14	420
Cavity Frame	5.82	295

2.3.2. Riser Ducts

In continuum limits, the fluid flow is governed by the Navier-Stokes equations. The fluid properties are assumed constant, yielding the incompressible Navier-Stokes equations. Heat transfer is also accounted for in the system; therefore, the heat conservation equation is added to the governing equations for the thermal-hydraulic model:

$$\nabla \cdot \mathbf{u} = 0 \tag{2a}$$

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla P + \nabla \cdot \tau + \rho \mathbf{f} \tag{2b}$$

$$\rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) = \nabla \cdot (k \nabla T) + \dot{q} \tag{2c}$$

where t is time, $\mathbf{u}$ is the velocity vector, ρ is density, P is pressure, τ is the viscous stress tensor, $\mathbf{f}$ is a force vector, C_p is the isobaric specific heat capacity, T is temperature, k is the thermal conductivity, and $\dot{q}$ is a volumetric heat source. The properties are assumed to remain constant within the estimated temperature range of the system.

The bouyancy force was modeled using the Boussinesq approximation, which assumes that the fluid density changes linearly with sufficiently small temperature differences in the system. Density variations were then neglected in the Navier-Stokes equations but accounted for in the body force gravity term. The Boussinesq approximation can be expressed as:

$$\rho = \rho_0 - \beta \rho_0 \Delta T \tag{3}$$

where β is the thermal expansion coefficient of the system.

The system's governing equations were solved using NekRS. LES was employed for turbulence modeling, incorporating an explicit filter to replicate the dissipation effect of the sub-grid scales. The numerical model is depicted on the left in Figure 3. The numerical models of the riser ducts were constructed based on the specifications outlined in the experimental report [2].

Figure 3 provides an overview of both the computational mesh and the employed boundary conditions. As indicated, a recycling region was implemented at the inlet to ensure a fully developed flow enters the riser ducts. The outflow treatment presented in [17] was applied at the outlet to avoid the backflow issue. Lastly, the duct walls assume a no-slip condition.

The data in the risers from the experimental report are limited solely to temperature measurements. Thus, a velocity recycling zone was added to the geometry to provide the riser ducts with a fully developed velocity profile. This enabled us to calculate pressure drop by using correlations in the literature, and to provide pressure validation for the system.

A Dirichlet boundary condition function was employed for the energy equation (i.e., 2c). This function was derived from the thermocouple measurements on the front surface of the riser wall. (The function can be seen on the right in Figure 3.) The thermal-hydraulic model was intended to focus on predicting the temperature difference at the center of the flow between the inlet and outlet.

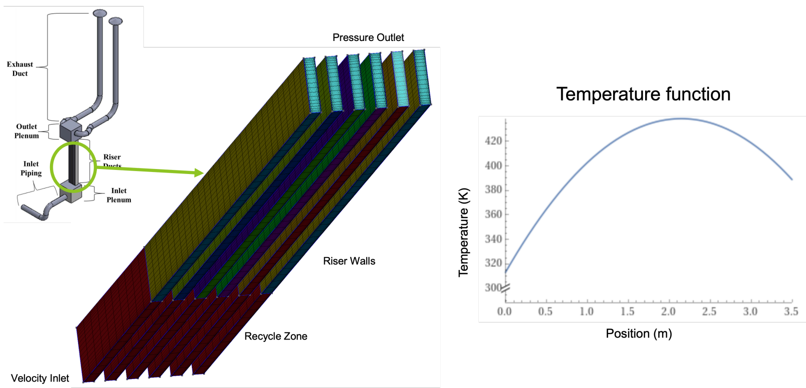

Figure 3. NekRS model of the risers (left) and temperature function for the riser walls (right).

3. RESULTS

This section focuses on providing the validation of initial results, by way of developing a numerical benchmark for the air-cooled RCCS experiment detailed in the previous sections. Subsection 3.1 focuses on the results of the thermal model created using the heat transfer module in MOOSE. Subsection 3.2 presents high-fidelity results generated by NekRS for the riser ducts.

3.1. Heated Cavity

Figure 4 presents the temperature contours produced by the heated cavity setup for test #14. The axial cut in the middle of the domain is taken to show temperature distribution within the cavity. The temperature profile shows consistent results with the expectations for the wall of the risers. The insulation material was added to the frame in order to reduce the amount of heat loss to the environment.

For a more quantitative comparison between the experiment and the MOOSE model, the axial temperature distributions for the front surface of the fourth riser were compared. The results were largely in agreement with each other, especially within the middle parts of the domain. However, certain inconsistencies observ-

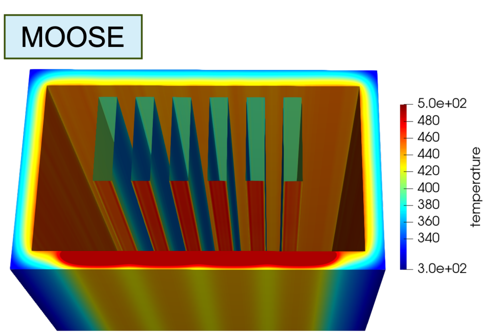

Figure 4. Temperature (K) contour of the heated cavity as axial cut in the middle of the domain.

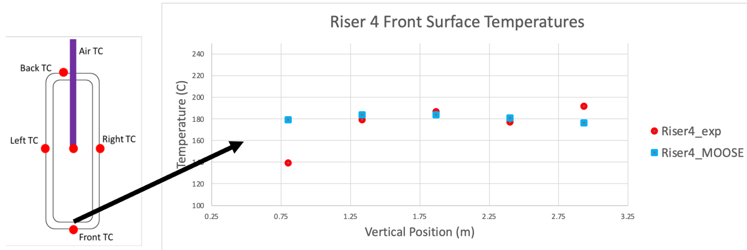

Figure 5. Temperature comparison between MOOSE and the RCCS experiment [2].

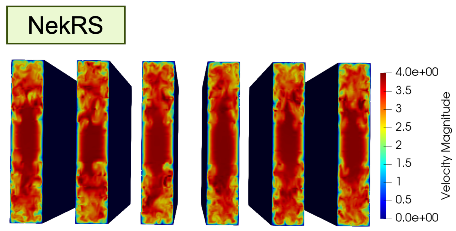

Figure 6. Velocity (m/s) contour of the riser duct.

able closer to the lower and upper ends of the domain may have been caused by using a constant heat transfer coefficient and bulk temperature throughout the entire domain. More sophisticated models for calculating heat transfer coefficients or modeling the fluid inside the cavity could serve to improve the results.

3.2. Riser Ducts

The riser ducts were simulated by using the NekRS code in tandem with the LES methodology. The Reynolds number (Re) for the system was estimated to be 15,615, indicating highly turbulent flow for the elongated duct geometry. Two cases isothermal and non-isothermal were run to simulate the thermal hydraulics of the riser ducts. The former case validated the hydraulic characteristics; the latter focused on expanding the validation of the thermal hydraulics by employing experimental data for the riser ducts.

For the isothermal case, Figure 6 visualizes the flow, from the axial view. As expected, the results show a consistent velocity profile with the duct geometry. Turbulent structures often appear closer to the wall region, due to the strong shear force.

3.2.1. Pressure Validation

The isothermal case was intended to to provide hydraulic validation of the numerical model constructed for the riser ducts. In this section, the Navier-Stokes equations reflected in Equations 2a and 2b are solved, without including the energy equation. To ensure the pressure profile was developed, the pressure drop along the fourth riser duct was calculated once two flow-through cycles had been completed after the initial start. To validate the numerical model, the same pressure drop was calculated using the following correlations: Bhatti and Shah [18], McAdams, and Colebrook [19]. The flow conditions satisfied the validity range of these correlations. The results are given in Table III, which compares the pressure drop estimated by NekRS and the correlations. The difference between the simulation and correlation results is less than 5%, clearly indicating that the numerical model can predict the correct pressure profile. The comparison with the Bhatti and Shah correlation is more physical in this case, since that specific correlation was established for elongated ducts, whereas the others were developed for pipe flow.

Table III. Pressure comparison between NekRS and the literature.

Method	Pressure Drop (Pa)	Difference
NekRS	5.71	-
Bhatti & Shah	5.51	3.5%
McAdams	5.78	1.22%
Colebrook	5.98	4.64%

3.2.2. Temperature Validation

The non-isothermal case focused on the validation of the riser duct system, using the available experimental data. The energy equation was added to the system and buoyancy modeled using the Boussinesq approximation. The same Re number given in the previous section was again used here as well. To allow the temperature profile to develop, the simulation was run for a couple of flow throughs before we began averaging the flow field. The temperature field statistics were then collected for two flow throughs in order to estimate the averaged temperature profile.

Fig. 7 presents the axial temperature profile that was obtained with NekRS when using 3th and 5th polynomial orders, and compares them against the experiment results. The experiment results proved a good match with the NekRS results generated when using the 5th polynomial order. Additional simulations will be run with higher polynomial orders to ensure that the results are independent of the mesh. The flow will be averaged for more time to ensure that the results are statistically converged.

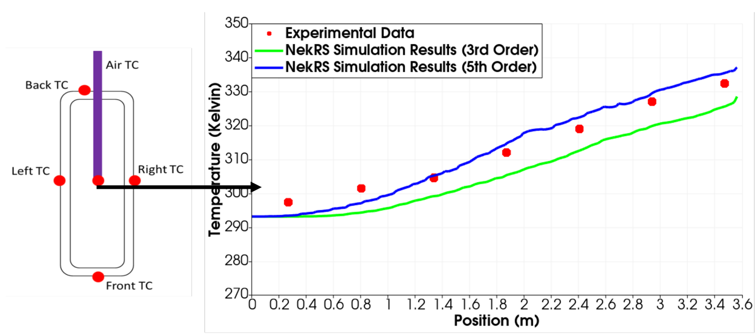

Figure 7. Temperature comparison between NekRS and the experiment.

4. CONCLUSION

The present work lays the groundwork for a broader effort aimed at establishing a numerical benchmark based on the experimental facility at UW-Madison [2]. Here, we present results pertaining to the initial validation of the tools to be employed in the benchmarking effort. This validation encompasses fundamental aspects of the modeling strategy. Specifically, the models developed in this work showcase the strategy for simulating the main physical aspects of the RCCS, including buoyancy effects, radiation heat transfer, and turbulence.

Two main models are presented in this work. First, we developed a MOOSE thermal model based on a tall cavity correlation, which is reminiscent of an RCCS air cavity. This model features a fast-running numerical tool for evaluating the overall heat transfer in the cavity. The second model affords a high-fidelity simulation of the riser ducts in the RCCS system. These models were validated using available data in the literature and/or experimental results from UW-Madison.

Ongoing efforts are being made to improve the simple MOOSE thermal model by explicitly modeling the air inside the cavity, thus allowing for more representative heat transfer coefficients. Future endeavors will focus on combining the initially developed models in order to perform multiphysics, high-fidelity simulations of the full model of the experimental facility, including all its components.

ACKNOWLEDGEMENTS

This research was funded by the U.S. Department of Energy (DOE)'s Advanced Reactor Program for Gas Cooled Reactors (ART-GCR). This research made use of Idaho National Laboratory computing resources, which are supported by the DOE Office of Nuclear Energy and the Nuclear Science User Facilities, under contract no. DEAC07-05ID14517.

REFERENCES

[1] J. M. Ryskamp, E. J. Gorski, E. A. Harvego, S. T. Khericha, and G. A. Beitel. "New Generation Nuclear Plant – High Level Functions and Requirements." Technical report, Idaho National Laboratory (2003). URL https://www.osti.gov/biblio/910744.

[2] M. Corradin, M. Anderson, M. Muci, Y. Hassan, A. Dominguez, A. Tokuhiro, and K. Hamman. "Thermal-Hydraulic Analysis of an Experimental Reactor Cavity Cooling System with Air. Part I: Experiments; Part II: Separate Effects Tests and Modeling." Technical report, Argonne National Laboratory (2014). URL https://www.osti.gov/biblio/1183658.

[3] D.-H. Shin, C. S. Kim, G.-C. Park, and H. K. Cho. "Experimental analysis on mixed convection in reactor cavity cooling system of HTGR for hydrogen production." *International Journal of Hydrogen Energy*, **volume 42**(34), pp. 22046–22053 (2017). URL https://www.sciencedirect.com/science/article/pii/S0360319917326757.

[4] F. P. Incropera and D. P. DeWitt. *Fundamentals of Heat and Mass Transfer*. John Wiley & Sons, Inc., New York City, New York, 4th edition edition (1996).

[5] P. M. Bardet and P. F. Peterson. "Options for Scaled Experiments for High Temperature Liquid Salt and Helium Fluid Mechanics and Convective Heat Transfer." *Nuclear Technology*, **volume 163**(3), pp. 344–357 (2008). URL https://doi.org/10.13182/NT163-344.

[6] D. D. Lisowski, C. D. Gerardi, R. Hu, D. J. Kilsdonk, N. C. Bremer, S. W. Lomperski, A. R. Kraus, M. D. Bucknor, and M. T. Farmer. "Water NSTF Design, Instrumentation, and Test Planning." Technical report, Argonne National Laboratory (2017). URL https://www.osti.gov/biblio/1375452.

[7] D. D. Lisowski, C. D. Gerardi, D. J. Kilsdonk, N. C. Bremer, S. W. Lomperski, R. Hu, A. R. Kraus, M. D. Bucknor, Q. Lv, T. Lee, and M. T. Farmer. "Final Project Report on RCCS Testing with Air-based NSTF." Technical report, Argonne National Laboratory (2016). URL https://www.osti.gov/biblio/1350591.

[8] R. Hu, A. Kraus, M. Bucknor, Q. Lv, and D. Lisowski. "Final Project Report on Computational Modeling and Analysis of Air-Based NSTF." Technical report, Argonne National Laboratory (2016). URL https://www.osti.gov/biblio/1429403.

[9] R. Freile, M. Tano, P. Balestra, S. Schunert, and M. Kimber. "Improved natural convection heat transfer correlations for reactor cavity cooling systems of high-temperature gas-cooled reactors: From computational fluid dynamics to Pronghorn." *Annals of Nuclear Energy*, **volume 163**, p. 108547 (2021). URL https://www.sciencedirect.com/science/article/pii/S0306454921004230.

[10] R. Hu, D. D. Lisowski, M. D. Bucknor, A. R. Kraus, and Q. Lv. "Ambient and building condition effects modeling of air-cooled natural circulation systems." *Applied Thermal Engineering*, **volume 137**, pp. 23–31 (2018). URL https://www.sciencedirect.com/science/article/pii/S1359431117371314.

[11] D.-H. Shin, S.-Y. Kim, C. S. Kim, G.-C. Park, and H. K. Cho. "Study on the Turbulent Mixed Convection Phenomena Inside the Air-Cooled RCCS Riser." In , volume Volume 6B: Thermal-Hydraulics and Safety Analyses of *International Conference on Nuclear Engineering*, p. V06BT08A029 (2018). URL https://doi.org/10.1115/ICONE26-82071.

[12] A. D. Lindsay, D. R. Gaston, C. J. Permann, J. M. Miller, D. Andrš, A. E. Slaughter, F. Kong, J. Hansel, R. W. Carlsen, C. Icenhour, L. Harbour, G. L. Giudicelli, R. H. Stogner, P. German, J. Badger, S. Biswas, L. Chapuis, C. Green, J. Hales, T. Hu, W. Jiang, Y. S. Jung, C. Matthews, Y. Miao, A. Novak, J. W. Peterson, Z. M. Prince, A. Rovinelli, S. Schunert, D. Schwen, B. W. Spencer, S. Veeraraghavan, A. Recuero, D. Yushu, Y. Wang, A. Wilkins, and C. Wong. "2.0 - MOOSE: Enabling massively parallel multiphysics simulation." *SoftwareX*, **volume 20**, p. 101202 (2022). URL https://www.sciencedirect.com/science/article/pii/S2352711022001200.

[13] A. G. Weiss, L. J. Zaidan, M. H. Bani Ahmad, M. G. Abdoelatef, J. W. Peterson, A. D. Lindsay, F. Kong, K. Ahmed, and M. L. Kimber. "Characterization of the Finite Element Computational Fluid Dynamics Capabilities in the Multiphysics Object Oriented Simulation Environment." *Journal of Nuclear Engineering and Radiation Science*, **volume 9**(2) (2022).

[14] P. Fischer, S. Kerkemeier, M. Min, Y.-H. Lan, M. Phillips, T. Rathnayake, E. Merzari, A. Tomboulides, A. Karakus, N. Chalmers, and T. Warburton. "NekRS, a GPU-accelerated spectral element Navier–Stokes solver." *Parallel Computing*, **volume 114**, p. 102982 (2022). URL https://www.sciencedirect.com/science/article/pii/S0167819122000710.

[15] E. Merzari, R. Rahaman, M. Min, and P. Fischer. "Performance Analysis of Nek5000 for Single-Assembly Calculations." In , volume Volume 2 of *Fluids Engineering Division Summer Meeting* (2018). URL https://doi.org/10.1115/FEDSM2018-83517. V002T09A031.

[16] R. K. MacGregor and A. F. Emery. "Free Convection Through Vertical Plane Layers—Moderate and High Prandtl Number Fluids." *Journal of Heat Transfer*, **volume 91**(3), pp. 391–401 (1969). URL https://doi.org/10.1115/1.3580194.

[17] S. Dong and J. Shen. "A pressure correction scheme for generalized form of energy-stable open boundary conditions for incompressible flows." *Journal of Computational Physics*, **volume 291**, pp. 254–278 (2015).

[18] M. Bhatti. "Turbulent and transition flow convective heat transfer in ducts." *Handbook of single-phase convective heat transfer* (1987).

[19] N. Todreas, M. Kazimi, and M. Massoud. *Nuclear Systems Volume II: Elements of Thermal Hydraulic Design*. CRC Press (2021). URL https://books.google.com/books?id=WttKEAAAQBAJ.