INL/CON-24-79127-Revision-0

Multi-physics Modeling of Radiative Heat Transfer and Fluid Flow for the Reactor Cavity Cooling System

November 2024

Changing the World's Energy Future

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

INL/CON-24-79127-Revision-0

Multi-physics Modeling of Radiative Heat Transfer and Fluid Flow for the Reactor Cavity Cooling System

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

November 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

Multi-physics Modeling of Radiative Heat Transfer and Fluid Flow for the 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, Idaho ;
[2]The Pennsylvania State University, State College, Pennsylvania

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

ABSTRACT

High-temperature gas-cooled reactors (HTGRs) are notable for their high thermal efficiency and potential for combined heat and power applications. These reactors are particularly appealing due to their advanced passive safety features. HTGRs utilize passive safety systems that function without requiring active components like pumps or compressors during emergencies. These reactor designs depend on a Reactor Cavity Cooling System (RCCS) to manage decay heat removal from the reactor pressure vessel (RPV) during accident conditions. The RCCS consists of vertical rectangular channels known as "risers" or riser ducts positioned around the RPV. These risers receive heat from the RPV through both convective and radiative heat transfer mechanisms. Understanding the interplay of multiple physical phenomena, such as fluid dynamics, heat transfer, and neutron interactions, is essential for the effective design and operation of nuclear reactors, particularly for systems like the RCCS. Multi-physics simulations provide a comprehensive approach to studying these interactions, offering detailed insights and enhancing accuracy. They are especially important in RCCS designs, where the interaction between radiative and convective heat transfer can significantly impact system performance. By leveraging multi-physics simulations, complex reactor behaviors can be modeled without compromising the fidelity of the underlying physical processes. This work aims to establish a robust methodology for coupling multiple physical processes in an air-cooled RCCS. By focusing on validating this multi-physics approach, the study involves designing test cases that simulate various conditions to verify the numerical models employed. The outcomes of this research will provide critical insights for accurately modeling and optimizing complex nuclear systems like the RCCS.

Keywords: Multi-physics, Radiative heat transfer, Cardinal, Reactor Cavity Cooling System, NekRS

1. INTRODUCTION

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 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 principles of nature, do not employ compressors, pumps, or other active components during accident scenarios. This means minimal 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 decay heat removal over long-term transients from the reactor pressure vessel (RPV) in accident scenarios. The RCCS includes vertical

*sinan.okyay@inl.gov

rectangular ducts called ”risers” (or riser ducts) around the RPV. These risers receive the heat from the RPV through convection and radiation heat transfer[3].

Many applications in nuclear reactors (including RCCS designs) are governed by the interaction of various physical processes (such as fluid flow, heat transfer, and neutron transport). Vendors must understand these interactions to ensure these systems meet their design specifications, particularly during accidents. Multi-physics simulations offer deeper insights into the interactions between various physical phenomena, leading to more reliable and accurate predictions and optimized designs. In RCCS designs, radiative and convective heat transfer strongly influence one another. This influence becomes even more pronounced since the system relies on natural circulation. Multi-physics simulations can be utilized to demonstrate complex systems without compromising the accuracy of the underlying physics. When validated by experimental data, they provide valuable insights and enhance the understanding of complex physical systems.

Recent advancements in the thermal-hydraulic characterization of Reactor Cavity Cooling Systems (RCCS) have been driven by numerous experimental and numerical studies, as highlighted in works such as [2, 4, 5, 6, 7]. An essential contribution to this field has been made by the Argonne National Laboratory (ANL) through their Natural Convection Shutdown Heat Removal Test Facility (NSTF). This facility has provided experimental and numerical data crucial for modeling the RCCS designed for General Atomics' Modular High-Temperature Gas Reactor (GA-MHTGR) [4, 5, 6]. Moreover, computational fluid dynamics (CFD) tools have been employed to enhance the performance of system-level thermal-hydraulic analysis codes, as documented in [6, 7]. However, to the best of our knowledge, a notable gap exists in the literature concerning high-fidelity simulations of the RCCS. Combining multi-physics tools with high-fidelity approaches promises to deliver more accurate and reliable numerical data, thereby improving the understanding of the thermal-hydraulic characteristics of RCCS designs.

This work aims to develop and validate a robust high-fidelity methodology for multi-physics coupling in an air-cooled Reactor Cavity Cooling System (RCCS). The study emphasizes the thermal-hydraulic analysis, designing test cases to verify the multi-physics methodology and ensure the accuracy and reliability of the numerical tools employed. The results will provide critical insights for accurately modeling and optimizing complex nuclear systems like the RCCS. Additionally, this research contributes to establishing a benchmark for RCCS designs, advancing the understanding and safety of nuclear reactors. We envision that the results will be used to improve faster running methods based on systems analysis codes or other engineering tools.

2. METHODS

This section outlines the methodology employed in the present study. The RCCS scenario is briefly explained, followed by a description of the modeling strategy, including the governing equations. Finally, the numerical tools used in this study are introduced.

2.1. RCCS Facility

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. Electrical resistance heaters simulate the RPV within the heated cavity and emit heat to the six riser ducts. Finally, the outlet plenum promotes mixing before releasing the hot air into the environment through a pair of exhaust ducts. The orange arrows in Figure 1 indicate the airflow direction inside the RCCS.

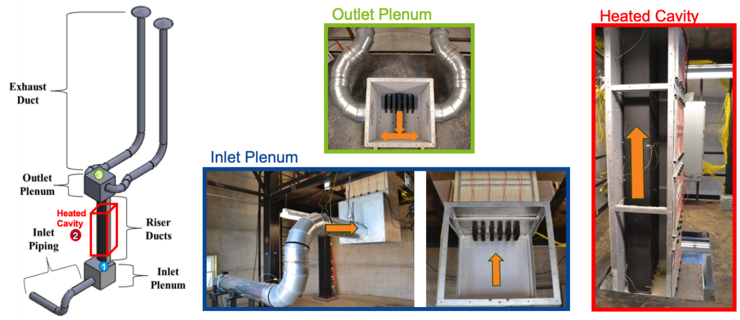

Figure 1. Facility overview (images taken from [2]).

2.1.1. RCCS Heated Cavity

The Reactor Cavity Cooling System (RCCS) receives heat from the Reactor Pressure Vessel (RPV) through various heat transfer mechanisms. The heated cavity is crucial to the air-cooled RCCS facility, serving as a thermal enclosure that transfers heat from the heaters (which act as the RPV) to the six riser ducts while minimizing heat losses to the environment. Similar to the reactor cavity of the GA-MHTGR, natural convection cells develop within the heated cavity. Heat transfer occurs primarily by radiation and secondarily by convection. Additionally, conduction occurs between the heaters and the frame of the heated cavity as the heaters are mounted within the cavity frame. Figure 2 presents a heat diagram of the heated cavity system. It should be noted that in Figure 2, the heat transferred to the frame is lost to the environment.

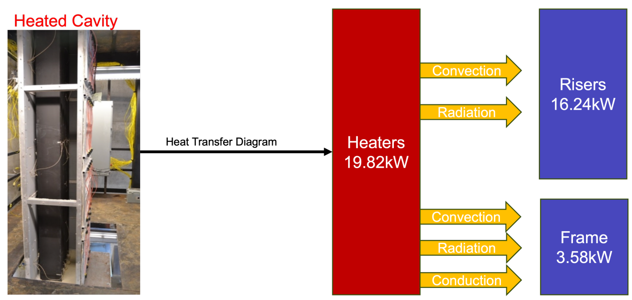

Figure 2. RCCS heat transfer diagram.

The heated cavity of the RCCS is the primary focus of the modeling strategy discussed in this study. The following sections will elaborate on the governing equations and the modeling approaches employed. Subsequently, the numerical models used to evaluate these strategies will be introduced.

2.2. Governing Equations & Multi-physics Coupling Strategy

2.2.1. Solid Heat Transfer

The energy transfer in the solid domain is dictated by the heat conduction equation. The heat conduction equation can be seen below.

$$\rho C_p \frac{\partial T}{\partial t} - \nabla \cdot (k_{solid} \nabla T) = 0 \tag{1}$$

here, T is the temperature and k_{solid} is the thermal conductivity of the solid. The time-dependent term will be neglected in the solid energy equations since the study focuses on steady-state analysis. The test cases do not include a heat source term; instead, heat is supplied as a flux at one of the boundaries. In any case, the final model in the heated cavity will include a source term that is equally distributed on the heaters.

2.2.2. Fluid Thermal-Hydraulics

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 of the working fluid, T is temperature, and k is the thermal conductivity of the fluid. The properties are assumed to remain constant within the estimated temperature range of the system. The buoyancy force was modeled using the Boussinesq approximation, which assumes that the fluid density changes linearly with sufficiently small temperature differences in the system. Based on the reference facility, air is employed as the working fluid.

2.2.3. Radiative Heat Transfer Modeling

Radiative heat transfer is modeled using the net radiation method [8]. This method assumes that the domain does not participate in heat transfer and heat exchanges between gray diffusive surfaces. It provides a robust calculation of radiative heat transfer. This approach is particularly valid in this case since air is not an opaque fluid and is not expected to participate in radiative heat transfer. A study [9] confirms this assumption in a canonical case (tall cavity) similar to a heated cavity scenario. The radiative heat flux for a surface is calculated using the formula below.

$$q'' = J_i - \sum_{j=1}^{N} F_{ij} J_j \tag{3}$$

where J_i and J_j are the radiosities at reference locations i and j, N is the total number of participating surfaces, and F_{ij} is the view factor between locations i and j. Further, the radiosity can be calculated using the formula below.

$$J_i = \epsilon_i \sigma T_i^4 + (1 - \epsilon_i) \sum_{j=1}^{N} F_{ij} J_j \tag{4}$$

There are three unknowns in the net radiation equations; however, there are only two equations to describe the system. The last equation depends on the boundary conditions of the modeled system. The system can be solved for either known surface temperatures or if the surface is adiabatic. This study will use the known surface temperature conditions, and the temperatures will be obtained through the coupling strategy introduced in the next section . The last equation is derived from the boundary condition.

$$\sum_{j=1}^{N}(\delta_{ij} - (1 - \epsilon_i)F_{ij})J_i = \epsilon_i \sigma T_i^4 \quad (5)$$

where δ_{ij} is the Kronecker delta function, which is one if i equals j and zero otherwise.

2.2.4. Multi-physics Coupling Strategy

Figure 3 presents the multi-physics coupling strategy. The strategy consists of three domains to improve the stability of each non-linear system: Heat Conduction, Nek Fluid Flow (Convection), and Radiation (the solid, fluid, and radiation systems are shown in red, blue, and yellow colors, respectively, in Figure 3). These systems are coupled through their boundary conditions. Variables such as temperature and heat flux are transferred as fields using nearest-location transfer and their integrated values (total power) to maintain heat balance between the systems.

The time step starts with solving heat conduction in the solid domain and then proceeds to convection in the fluid flow. Finally, the radiative heat transfer between the surfaces of the fluid domain is solved. These time steps are repeated until the solution converges to a steady-state value. The fluid flow requires smaller time steps; therefore, for each solid heat conduction time step, several time steps are performed for fluid flow. This method is also called sub-cycling [10]. The boundary conditions of the fluid flow are dependent on radiative heat transfer, so the radiative heat transfer is calculated for each fluid time step.

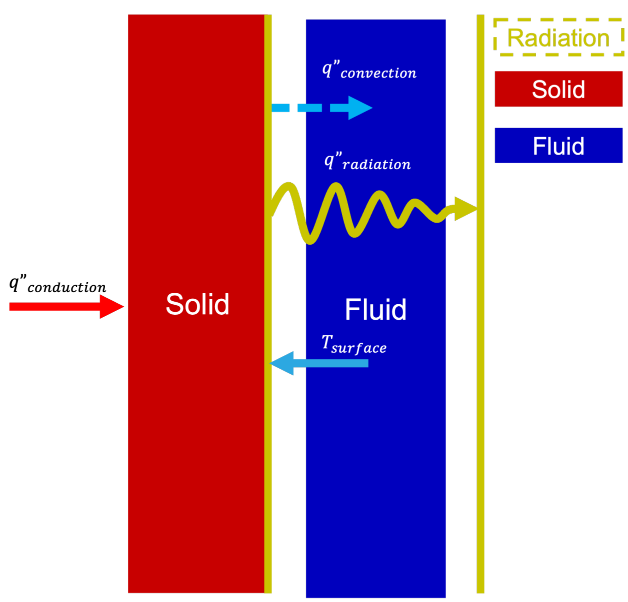

Figure 3. Multi-physics Coupling Methodology.

The process begins by solving the heat conduction equation in the solid domain. The conductive heat flux and total power (integral of the conductive heat flux) are then transferred to the fluid region. The solid region receives the temperature at the interface as a boundary condition from the fluid region.

Next, the fluid simulation uses the heat flux as a boundary condition to solve fluid thermal-hydraulics and calculates the temperatures within the domain. The surface temperatures are then transferred to the radiation module, which simulates the radiative heat transfer using the net radiation method. The radiation module calculates the heat flux transfer between surfaces based on the surface temperatures from the fluid domain. Since the heat is transferred between the surfaces by radiative heat transfer, the convective heat flux is adjusted to maintain heat balance. This adjustment ensures that the total convective power equals the total power (coming from conduction) minus the radiative power (integral of radiative heat flux).

2.3. 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 the Cardinal tool from the program [11]. Cardinal can wrap the spectral element code NekRS for Computational Fluid Dynamics (CFD) and the MOOSE framework applications, enabling the large-scale, first-of-a-kind simulation of energy systems.

MOOSE is a framework for performing multi-scale and multi-physics simulations developed by Idaho National Laboratory (INL) [12]. The MOOSE framework implementation primarily relies on the finite element method (FEM). NekRS is an open-source CFD solver that utilizes the spectral element method (SEM). It is employed by the NEAMS program to perform advanced nuclear reactor thermal-hydraulic analyses at various fidelity levels [13].

The RCCS will be simulated as a conjugate heat transfer case. Heat transfer in the solid will be modeled using the MOOSE tools, incorporating both conduction and radiative heat transfer. Fluid thermal hydraulics will be modeled using NekRS. The radiation will be modeled using the net radiation method described above, with view factors calculated using the Ray-tracing module of the MOOSE Framework.

3. RESULTS

This section presents the results of our study, divided into two parts. The first one focuses on the canonical case setups, discussing the verification of the NekRS numerical tool and results of the test cases for the multi-physics methodology. This includes a detailed analysis of the riser ducts, a crucial component of RCCS designs, and simple canonical setups demonstrating the multi-physics approach. The second subsection presents the preliminary thermal-hydraulic results of the RCCS case and discusses the findings from these results.

3.1. Results for Canonical Cases

This section will present the results for the canonical cases. The following section will focus on verifying the NekRS numerical tool. Then, the test cases for the multi-physics methodology will be introduced.

3.1.1. Riser Ducts

The riser ducts are the main components of the RCCS designs. Their hydraulic characteristics have crucial effects on the performance of RCCS. The riser ducts are responsible for a significant portion of the pressure drop in the RCCS, mainly caused by viscous losses. They have the hydraulic characteristics of the elongated ducts. The geometry is presented in Figure 4.

The riser ducts were simulated using the NekRS code in tandem with the Large Eddy Simulations (LES) methodology. LES was employed for turbulence modeling, incorporating an explicit filter to replicate the dissipation effect of the sub-grid scales. The Reynolds number (Re) for the system was estimated to be 15,615, indicating highly turbulent flow for the elongated duct geometry. The numerical models of the riser ducts were constructed based on the specifications outlined in the experimental report [2].

The case was intended to provide hydraulic verification of the numerical model constructed for the riser ducts. Therefore, the case is assumed to be isothermal. Figure 4 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 [14] was applied at the outlet to avoid the backflow issue. Lastly, the duct walls are assumed to have a no-slip condition.

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.

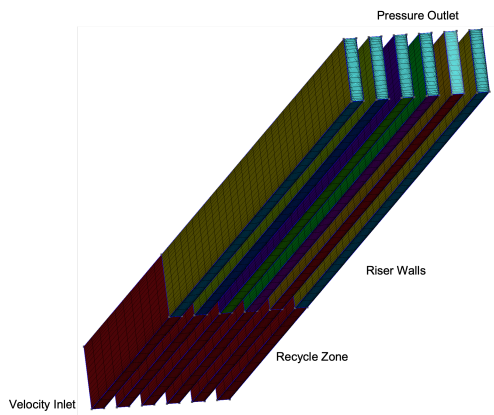

Figure 4. NekRS hydraulic model setup.

To verify the numerical model, the same pressure drop was calculated using the following correlations: Bhatti and Shah [15], McAdams and Colebrook [16]. The flow conditions satisfied the validity range of these correlations. The results are given in Table I, 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 drop. 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. Additional analyses were performed to ensure the reliability of the results. Simulations were performed at different polynomial orders in the NekRS solver, yielding nearly identical results and thus ensuring grid independence. Moreover, the solution with a 6th-order polynomial has a resolution adequate to resolve the Taylor microscale, which is necessary to address all relevant scales of turbulence in LES [17].

Table I. 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%

3.1.2. Multi-physics Test Cases

This section presents results for two simple setups as a proof of concept for the multi-physics approach proposed. Two canonical models are proposed: a slab cooled by laminar flow and a radiative slab cooled by laminar flow. The numerical setups are shown in 5. The results obtained from the simulations will be compared with the theoretical values obtained from hand calculations for each case.

Solid and fluid domains are modeled as rectangular parallelepipeds. The dimensions of the solid domain are 0.25 m x 0.25 m x 1 m, while the dimensions of the fluid domain are 0.25 m x 0.25 m x 1.25 m. The additional height in the fluid region accounts for the inlet zone for fluid flow to improve stability. The fluid flow is laminar. The buoyancy force is not considered in this setup. The top and bottom walls in the solid region are assumed to be adiabatic. Heat is supplied as a Neumann boundary condition from the left side of the solid domain. The boundary conditions at the interface are Dirichlet, based on the surface temperatures from the fluid domain (the NekRS simulation). Velocity inlet and pressure outlet boundary conditions are used for the laminar fluid flow. The Reynold's number is calculated based on the hydraulic diameter of the square channel. The side walls have no-slip boundary conditions for the fluid. The properties of air are assumed to be constant under atmospheric conditions.

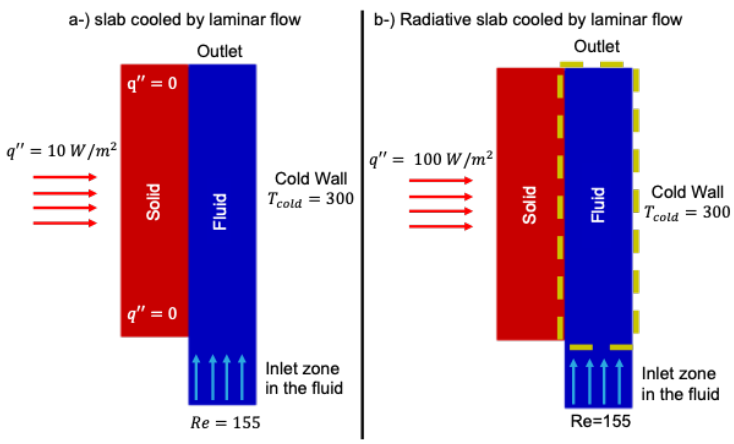

Figure 5. Numerical Setups for test cases.

The first numerical model is the heated slab cooled by laminar flow presented in Figure 5.a. This test case aims to demonstrate Cardinal's capabilities for a conjugate heat transfer case. The heated solid is cooled by a forced convection. The previously mentioned part of the multi-physics coupling strategy will be utilized for the first test case. Since there is no radiative heat transfer in the first case, only the transfers between heat conduction and fluid flow will be modeled. The analytical heat conservation equation is derived to calculate the theoretical values:

$$q''_{\text{conduction}} = q''_{\text{convection}} = 10\,\text{W/m}^2 \tag{6a}$$

$$-k_{\text{solid}}\frac{T_{\text{hot}} - T_{\text{interface}}}{s_{\text{solid}}} = h(T_{\text{interface}} - T_{\text{bulk}}) = 10\,\text{W/m}^2 \tag{6b}$$

where T_{hot} represents the temperature of the left boundary in the solid, $T_{\text{interface}}$ is the temperature at the interface between the solid and fluid, and s_{solid} is the thickness of the solid domain, which is 0.25 m. Additionally, k_{solid} represents the thermal conductivity of the solid, set at 0.01 W/mK, while h is the heat transfer coefficient of laminar flow calculated from a constant Nusselt number for laminar flow (Nu=4.36). The heat balance equation presented below is used to calculate the bulk temperature at the outlet.

$$Q_{Fluid} = \dot{m} C_p (T_{bulk,out} - T_{bulk,in}) \tag{7}$$

where Q_{Fluid} is the total energy transferred into fluid, $\dot{m}$ the mass flow rate, $T_{bulk,out}$ and $T_{bulk,in}$ is the outlet and inlet bulk temperatures.

The second canonical setup is the heated radiative slab cooled by a laminar flow. This test case includes more complicated physics than the first test case yet is simple enough to solve with hand calculations. The numerical setup can be seen in Figure 5.b.The radiative heat transfer is modeled in this numerical setup in addition to the first test case. Figure 5.b shows the radiative heat transfer boundaries as yellow dashed lines. The power value is increased for this case while the flow is kept in a laminar condition, which results in similar expected values in the fluid domain compared to the previous test case. This case aimed to be a radiative heat transfer dominated case because, in the RCCS scenario, radiative heat transfer is the significant form of heat transfer. The simplified form of the heat conservation equation is derived to calculate the theoretical values. The radiative heat transfer added to the heat balance equations, as seen in the equations below.

$$q''_{\text{conduction}} = q''_{\text{convection}} + q''_{\text{radiation}} = 100\,\text{W/m}^2 \tag{8a}$$

$$-k_{\text{solid}} \frac{T_{\text{hot}} - T_{\text{interface}}}{s_{\text{solid}}} = h(T_{\text{interface}} - T_{\text{bulk}}) \quad + \frac{\sigma\left(T^4_{\text{interface}} - T^4_{\text{cold}}\right)}{\frac{1}{\varepsilon_1} + \frac{1}{\varepsilon_2} - 1} = 100\,\text{W/m}^2 \tag{8b}$$

where σ is the Stefan-Boltzmann constant and ε_1 and ε_2 are the emissivity values of the surface of the interface and cold wall which are set to 0.8.

This model operates using the modeling strategy presented in Figure 3, which includes the transfers occurring with radiation. The theoretical and simulation values are compared in the table below for each case. The values calculated from Cardinal simulations show good agreement with the theoretical values. The small differences can be attributed to assumptions made in the hand calculations, such as assuming a constant Nusselt number along the interface and not accounting for spatial effects. Hand calculations do not include spatial effects (2-D or 3-D) that can arise from a grid; instead, they assume the domain as two points (0-D assumption). The last two columns show the power values calculated from the Cardinal simulations, showing how heat is distributed in each system.

A partial grid refinement study was conducted in the first test case (slab cooled by laminar flow) to ensure that the simulation results are not dependent on grid refinement. The number of divisions in the solid mesh was increased from 25 to 200 to observe the effect of grid refinement on the results. The results, shown in Figure 6, indicate that increasing the mesh divisions did not change the computed temperature values. The figure displays the outlet bulk temperature for various mesh divisions over time, demonstrating consistency across various refinements. One should note that the time in this figure represents the increase in the iterations since the analysis here is only focused on the steady-state results.

3.2. Preliminary Results for RCCS Case

This section presents the results for the RCCS cooling system based on the riser ducts model described previously. This model is more complex than the canonical setup, featuring realistic RCCS geometry and

Table II. Comparison of Theoretical, Cardinal Values, and Power Distribution

Slab cooled by laminar flow				
Temperature	**Theoretical Value**	**Cardinal Value**	**Power**	**Value**
$T_{bulk,out}$	303 K	303 K	$Q_{Convection}$	2.5 W
$T_{interface}$	322 K	319 K	$Q_{Radiation}$	N/A
T_{hot}	572 K	569 K	$Q_{Conduction}$	2.5 W
Radiative slab cooled by laminar flow				
Temperature	**Theoretical Value**	**Cardinal Value**	**Power**	**Value**
$T_{bulk,out}$	303 K	303 K	$Q_{Convection}$	2.36 W
$T_{interface}$	321 K	317 K	$Q_{Radiation}$	22.64 W
T_{hot}	2820 K	2817 K	$Q_{Conduction}$	25 W

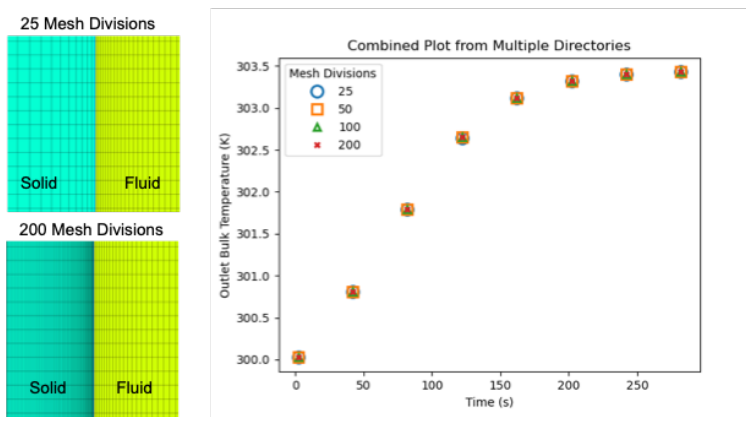

Figure 6. Grid independence study for multi-physics coupling approach.

incorporating the heat conservation equation in the fluid flow. It also accounts for buoyancy effects in the fluid. This initial model is designed to address the thermal-hydraulic aspects of the system. Temperatures at the riser duct walls are imposed as Dirichlet boundary conditions derived from experiments [2]. The final model will include the multi-physics strategy previously tested. Figure 7 shows the numerical model alongside a sketch of the experimental facility. The study's initial focus is to ensure accurate modeling of the inlet plenum, riser ducts, and outlet plenum. Therefore, this model does not yet include the exhaust ducts or all of the inlet piping.

This model contains more than 1.7 million elements, corresponding to 2.13 x 10^8 grid points for a 4th-order polynomial solution in NekRS. The preliminary results show turbulent jet interactions in the outlet plenum. The flow patterns observed here can significantly affect the performance of the RCCS. The following study [18] also observed that turbulent jet interactions can impact the system and change the flow characteristics. The instantaneous temperature at the outlet plenum is shown in Figure 8. The flow enters from the riser ducts, mixes in the outlet plenum, and then discharges through the exhaust ducts.

The current model requires further refinement to ensure consistent results in the fluid thermal-hydraulics part. The results are being post-processed to obtain averaged temperature, velocity, and pressure fields. The

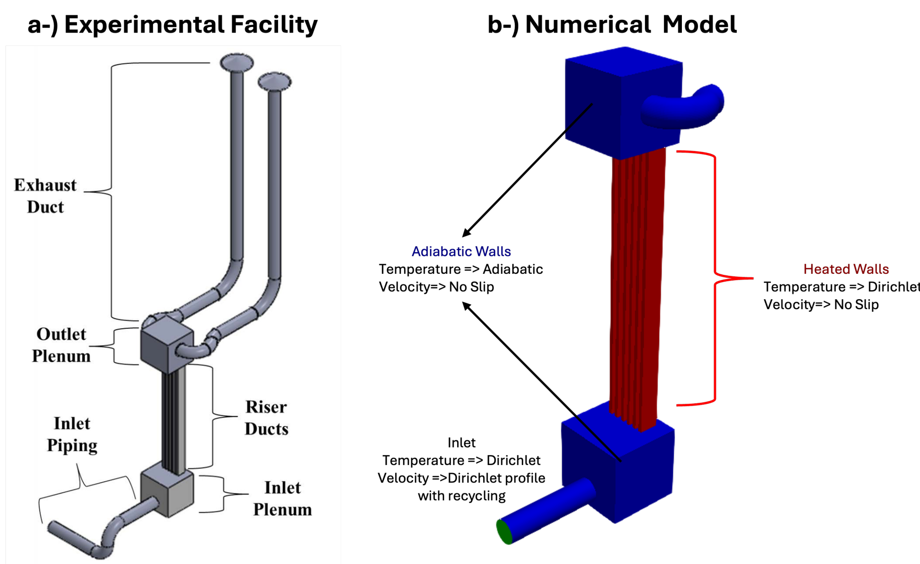

Figure 7. RCCS numerical setup.

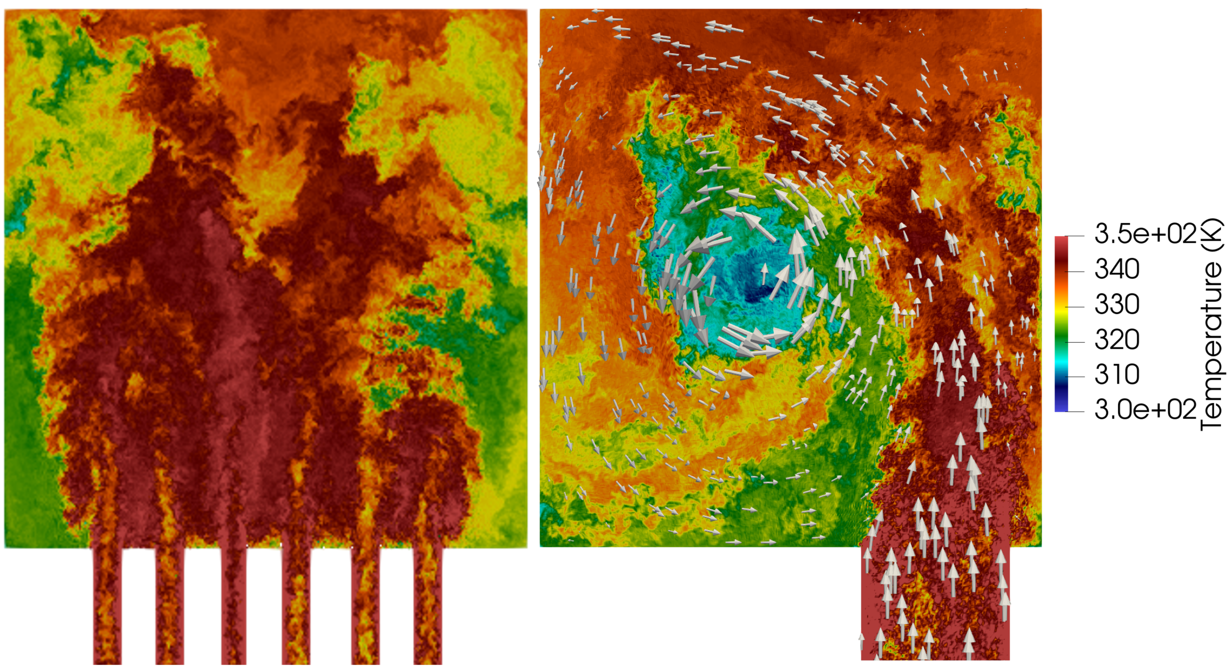

Figure 8. Instantaneous temperature at outlet plenum: left (front view) and right (side view). (White arrows were added to the side view to show the flow direction in the outlet plenum.)

temperature field will be compared to experimental data. Additionally, the model will be run with a higher polynomial order to ensure grid convergence of the results. The mesh analysis will include a comparison of grid length to Taylor's microscale to meet the length scale requirements for the LES methodology.

4. CONCLUSIONS

This study presents verification work for multi-physics simulations of an air-cooled reactor cavity cooling system. The first part of the results section focuses on canonical cases for verifying the numerical tools and the multi-physics coupling strategy.

The first canonical setup involves the riser ducts. Riser ducts are modeled as elongated duct flow to verify the NekRS numerical tool for the RCCS scenario. The simulation results show good agreement with the theoretical results calculated from various pressure drop correlations in the literature.

Next, the multi-physics coupling strategy is tested on two canonical setups. The initial test case demonstrates Cardinal's capabilities for a simple conjugate heat transfer scenario and verifies the grid independence of the coupling methodology.

The latter test case evaluates the model's capabilities under more complex physical conditions. Radiative heat transfer is modeled using a surface-to-surface method, and Cardinal's results are compared with theoretical values derived from hand calculations. The simulation results show good agreement with theoretical results. The methodology developed in this study will be used to model the RCCS (including the heated cavity scenario), as introduced in the methods section.

Preliminary results are presented for the realistic RCCS scenario. The results show turbulent jet interactions in the outlet plenum, which can significantly affect the heat removal characteristics of the RCCS. Asymmetric heating conditions can enhance these effects, causing partial blocking in the riser ducts or unequal flow redistribution in the exhaust ducts. Future models will analyze these preliminary findings to gain more information on the thermal-hydraulic characteristics of the RCCS design.

ACKNOWLEDGMENTS

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] 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.

[5] 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.

[6] 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.

[7] 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.

[8] 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).

[9] P. Kumar, G. Chanakya, and N. Bartwal. "Investigations of non-gray/gray radiative heat transfer effect on natural convection in tall cavities at low operating temperature." *International Communications in Heat and Mass Transfer*, **volume 125**, p. 105288 (2021). URL https://www.sciencedirect.com/science/article/pii/S0735193321001822.

[10] P. K. R. R. R. E. M. A. J. Novak, P. Shriwise and D. Gaston. "Coupled Monte Carlo Transport and Conjugate Heat Transfer for Wire-Wrapped Bundles Within the MOOSE Framework." *Nuclear Science and Engineering*, **volume 197**(10), pp. 2561–2584 (2023). URL https://doi.org/10.1080/00295639.2022.2158715.

[11] A. Novak, D. Andrs, P. Shriwise, J. Fang, H. Yuan, D. Shaver, E. Merzari, P. Romano, and R. Martineau. "Coupled Monte Carlo and Thermal-Fluid Modeling of High Temperature Gas Reactors Using Cardinal." *Annals of Nuclear Energy*, **volume 177**, p. 109310 (2022). URL https://www.sciencedirect.com/science/article/pii/S0306454922003450.

[12] C. Permann, D. Gaston, D. Andrš, R. Carlsen, F. Kong, A. Lindsay, J. Miller, J. Peterson, A. E. Slaughter, R. Stogner, and R. Martineau. "MOOSE: Enabling massively parallel multiphysics simulation." *SoftwareX*, **volume 11**, p. 100430 (2020).

[13] P. Fischer, S. Kerkemeier, M. Min, Y. 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).

[14] 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).

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

[16] 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.

[17] J. M. V. P. Victor Coppo Leite, Elia Merzari and A. Manera. "High-Fidelity Simulation of Mixing Phenomena in Large Enclosures." *Nuclear Science and Engineering*, **volume 198**(7), pp. 1386–1403 (2024). URL https://doi.org/10.1080/00295639.2023.2186159.

[18] E. Merzari, V. Coppo Leite, J. Fang, D. Shaver, M. Min, S. Kerkemeier, P. Fischer, and A. Tomboulides. "Energy Exascale Computational Fluid Dynamics Simulations With the Spectral Element Method." *Journal of Fluids Engineering*, **volume 146**(4), p. 041105 (2024). URL https://doi.org/10.1115/1.4064659.