MODELING THE THERMAL-HYDRAULIC BEHAVIOR OF THE REACTOR CAVITY COOLING SYSTEM USING RELAP5-3D

Authors: Rodolfo Vaghetto^a, Yassin A. Hassan^a

^aDepartment of Nuclear Engineering, Texas A&M University, College Station, TX 77843-3133,
USA

Contact Email Address: r.vaghetto@tamu.edu

Department of Nuclear Engineering

Texas A&M University

3133 TAMU

College Station, TX 77843-3133

Total number of pages: 30

Total number of Tables: 1

Total number of Figures: 13

ABSTRACT

The Very High Temperature Gas-Cooled Reactor (VHTR), one of the six proposed designs for the next generation nuclear reactor, was conceived to achieve high temperatures to support industrial applications and power generation. Due to the high temperature reached during normal operation, the design included new passive safety systems. The Reactor Cavity Cooling System (RCCS) is a new passive safety system designed to remove the heat from the reactor cavity during normal operation (steady-state) and accident scenarios. Computational tools such as system codes have been selected to simulate the reactor system and, in particular, the new safety components. The capabilities of these codes are being investigated to verify their ability in predicting the phenomena involved in the RCCS operation during steady-state and accident conditions. A RELAP5-3D input model of a small scale water-cooled experimental facility was prepared to simulate steady-state. The simulation results were compared with data produced during the experimental steady-state run. The results obtained and presented in the paper showed a good agreement of the code prediction with the experimental data. The paper also provides a set of modeling techniques to overcome some of the limitations of the current version of the computer code in simulating complex geometries with combined heat transfer mechanisms in the reactor cavity of the VHTR.

Keywords: Reactor Cavity Cooling System (RCCS); RELAP5-3D Modeling Techniques;

I. INTRODUCTION

The U.S. Department of Energy's Office of Nuclear Energy has engaged governments, industry, and the research community worldwide in the development of next generation nuclear energy systems known as "Generation IV." Technology goals of the Generation IV systems include safety and reliability, with the new systems' being characterized by a very low likelihood and degree of reactor core damage. Achieving increased safety and reliability will also require the introduction of new passive systems.

The Very-High-Temperature Reactor (VHTR) is a graphite-moderated, helium-cooled reactor designed to supply electricity and high-temperature (core outlet temperatures up to 1,000 °C) process heat to a broad spectrum of industrial processes such as hydrogen production or petrochemical manufacturing [1]. The reference reactor is a 600 MWth core with two configurations (prismatic block or pebble-bed).

Passive heat removal systems (based on natural circulation, for example) are one of the primary technological goals of the Generation IV program [2], since they can continue to function without human intervention even during a loss of site power. The Reactor Cavity Cooling System (RCCS) is a new safety system proposed for the VHTR that is designed to guarantee the integrity of the fuel, the reactor vessel, and the reactor cavity structures by removing heat leaked from the reactor vessel during both normal operation and accident scenarios.

The water-cooled RCCS design [2, 3] is a water-based cooling system that operates at low temperature and atmospheric pressure. Water temperatures are below 30°C during normal active operation, reaching the boiling point only during emergency passive operation. The

system can operate in active mode by removing the heat from the water in the tanks via an active secondary heat removal system, or in passive mode by boiling the water for approximately 72 hours. A small fraction of the heat produced in the core is released into the reactor cavity, mostly by radiation [4] and convection, and ultimately transferred to the RCCS water, flowing upward through vertical risers. Water coming from different risers is collected in horizontal headers or manifolds and then reaches the water tanks located on top of the reactor building, mixes with cold water, and comes back into the reactor cavity. The system is designed to guarantee the removal of approximately 0.7 MW during normal operation and up to 1.5 MW in case of accident [5].

The reactor cavity walls are lined with stainless steel vertical pipes (risers) that surround the reactor vessel. Figure 1 depicts a typical layout of the components inside the cavity of a VHTR.

Figure 1. Typical RCCS configuration.

The risers are organized in panels (red lines in Figure 1, right). The cooling system consists of 25 cooling panels, each containing nine risers. Since the majority of the heat in the cavity is transferred by radiation [4], the riser panels may include additional features to enhance the radiation heat transfer, such as stainless steel fins (welded along the side of two adjacent risers).

II. PURPOSE

The U.S. Department of Energy's (DOE's) Office of Nuclear Energy and the Nuclear Energy University Program (NEUP) fund research on the thermal-hydraulic behavior of the RCCS, its potential capability to remove the heat from the reactor cavity under different operating conditions, and the computational tools' ability to predict the phenomena involved in the RCCS. System codes such as RELAP5-3D have been selected to simulate a reactor system and safety components such as the RCCS.

A RELAP5-3D input model representing a small-scale water-cooled RCCS experimental facility was prepared to support the computer code verification, identify any weakness or limitations of the code models and features, and suggest specific modeling techniques to more accurately consider particular phenomena observed during the experimental activity.

III. FACILITY OVERVIEW AND DESCRIPTION

The experimental facility is a scaled (1:23) representation of the water-cooled RCCS simulating one of the several risers' panels that will be installed in the full-scale reactor. A detailed description of the facility and instrumentation installed is provided in [6].

A simplified scheme of the experimental facility layout is shown in Figure 2a. The main components of the facility are identified with numbers (1 to 8) and will be described in detail in the next paragraph.

Figure 2a. Experimental facility Scheme.

The experimental facility contains one cooling panel that would be used in an RCCS. This cooling panel consists of nine risers (1), two glass manifolds (2 and 3), and one water tank (4). The heat generated in the reactor vessel (6) (produced by three electrical radiant heaters) is transferred to the water flowing inside the risers (1). The increase in the water temperature brings about buoyancy forces that cause the water inside the system to circulate. Hot water is collected in the top manifold (3), flows through the upward pipe (7), and reaches the water tank (4). During the steady-state phase (corresponding to full-scale reactor nominal operation), a secondary heat removal system acts on the coolant in the water tank. Colder water moves downward through the downcomer (5) and enters the risers from the bottom manifold (2).

The manifolds (2 and 3) are connected to the risers' panel. One of the most remarkable features of the facility is the asymmetric layout of the cavity flow inlet and outlet. The coolant enters into the bottom manifold from the right end, and leaves the cavity through the top manifold's left end. This configuration, as will be described in the next section, was found to play an important role in the flow distribution through the risers.

A water tank (4) was installed at the top of the facility and connected to the manifold with vertical pipelines (5 and 7). All the components described above were thermally insulated in order to limit heat losses during the experiment. An overview of the experimental facility and its main components is presented in Figure 2b.

Figure 2b. Experimental facility Overview.

During the steady-state phase, the heat produced in the cavity and transferred to the water coolant, has to be removed from the water tank. A manually-operated apparatus was installed to remove the heat from the tank. Hot water collected in the water tank is removed by a small outlet located at the bottom of the tank. The water flows through a copper coil submerged in an ice container where the heat is removed by the ice's melting. Ice is introduced in small batches into the container during the steady-state to minimize the heat transfer perturbation. The introduction of the ice usually started when a desired temperature of the coolant in the downcomer is achieved (approximately 30°C). A variable-speed pump establishes the desired flow of water to maintain the steady-state conditions in the main system at a given downcomer water temperature. Colder water flows upward and it is injected into the water tank through a nozzle located at the top of the tank, where it mixes with the hot water. The facility is designed to operate at atmospheric pressure.

The instrumentation installed in the experimental facility includes thermocouples to measure the temperature of the cavity walls and coolant, and flow meters to measure the flow rates. In particular, k-type thermocouple probes were installed to measure the coolant temperature in different locations of the facility, such as cavity inlet (located at the entrance of the bottom manifold) and cavity outlet (located at the exit of the upper manifold).

The volumetric flow rate was determined using a magnetic flow-meter installed in the bottom section of the pipeline, upstream of the bottom manifold. An additional ultrasonic flow-meter was installed and used during the facility shakedown for comparison.

IV. RELAP5-3D MODEL PREPARATION AND PRELIMINARY ANALYSIS

The RELAP5 code has been developed for best-estimate transient simulation of LWR coolant systems during postulated accidents. RELAP5-3D [7] is a successor to the RELAP5/MOD3 code that was developed for the Nuclear Regulatory Commission. RELAP5-3D is one of the systems codes selected for the analysis of steady-state and transients required for the Generation IV nuclear power plants, including the HTGR [8].

A RELAP5-3D model of the experimental facility was prepared in order to conduct simulations of the RCCS facility during the steady-state phase.

A simplified nodalization diagram is presented in Figure 3.

Figure 3. RELAP5-3D nodalization diagram.

The risers were simulated with nine independent vertical pipe components (201-209). Five subvolumes were defined in each pipe, one at the center and thermally connected to the cavity by heat structures, and four (two at the top and two at the bottom) to simulate the portion of the risers outside the reactor cavity (and not connected to heat structures). The reactor cavity (space between the reactor vessel and the risers' panel) and the back cavity (area between the risers' panel and the cavity walls) were each simulated by one corresponding volume (not shown in Figure 3).

The upper manifold (225) and lower manifold (255) were simulated with horizontal pipe components with 15 subvolumes each. Components 201 to 209 were connected to manifolds 225 and 255 using multiple cross junctions, for which uniform form loss coefficients were specified. The water tank is simulated by a single volume (300).

The pressure boundary was imposed by connecting the top of the single volume 300 to a time-dependent volume for which atmospheric pressure and temperature were specified. Time-dependent volumes were also defined to impose the boundary conditions of the environment—to account for the cavity, pipes, and tank heat losses.

Sensitivity analyses were conducted to study the effect of the nodalization technique for specific components such as upper manifold and water tank. Simulations using different components (single volumes, pipes, multi-dimensional components) to simulate the upper manifold and the water tank were performed [9, 10]. For the purpose of the simulations presented in this paper, the nodalization selected is the one previously described.

The heat removal system in the water tank was modeled using a simplified nodalization with two time-dependent volumes (310 and 320), one time-dependent junction (305), and one single-junction (315). The thermal-hydraulic conditions (pressure and temperature) of the cold water from the ice container exchanger were imposed on time-dependent volume 310. These conditions were recorded during the experiment. The flow rate of the secondary coolant was imposed on time-dependent junction 305. Because minor adjustments were made to the pump speed during the experiment, a constant flow rate was imposed in the simulation. Junction 305 was connected to the top of component 300, simulating the water tank to faithfully represent the location of the water inlet in the tank. Atmospheric pressure and room temperature were imposed on the discharge volume (component 320). This simplified configuration allowed accounting for the heat removal of the secondary system without modeling the ice container heat exchanger.

Several heat structures were defined in the input deck to account for the complex heat transfer mechanisms expected in the cavity and for the heat losses between the experimental facility and the environment through walls and thermal insulation. Heat structures were also defined to account for the thermal losses of the pipelines and tank. The left boundary of the heat structures was connected to the hydrodynamic components that model the pipeline between the cavity and the water tank. The right boundary was connected to the single volume simulating the environment.

The organization of the heat structures in the reactor cavity required assumptions and additional calculations, due to the complexity of the geometry, the combined heat transfer

mechanisms expected to occur in the reactor cavity, and known constraints of the RELAP5-3D code when modeling radiation and conduction within the same heat structures. Figure4 (left) shows the top view of the reactor cavity nodalization. The heating panel (reactor vessel), the reactor cavity and back cavity (components 100 and 400), standing pipes and fins can be easily recognized in the figure.

Figure 4. Cavity Nodalization and Heat Structures.

Because nine independent pipe components were used to simulate the risers, the front wall of the pipes and the fins (exposed to radiation and convection heat transfer) were modeled using nine independent heat structures, each accounting for the front half of the pipe wall and two halves of the fin (left and right side of the pipe), as shown in Figure4 (right).

Figure 5 shows a simplified view of the heat transfer mechanisms between the vessel and the risers and the single parts of the heat structures involved.

Figure 5. Heat Transfer Mechanisms in the Cavity.

Heat is transferred from the reactor vessel (dark block at the bottom of the diagram in Figure 5) to pipes and fins by radiation (red lines) and by natural convection (blue lines) of air within the reactor cavity. The heat transferred to the fins is then transferred by conduction (horizontal red lines) to the pipes' wall and eventually by convection to the water flowing inside the pipes. The heat structures defined in RELAP5-3D can be grouped into radiation or conduction enclosures to simulate heat transfer. Because a given heat structure in RELAP5-3D cannot be specified in more than one enclosure, specific assumptions were necessary in order to overcome this limitation and produce an accurate modeling of the cavity heat transfer mechanisms. Previous publication [4] and experimental work conducted in similar RCCS facilities [11] have shown that radiation is the predominant heat transfer mechanism in the reactor cavity, carrying almost 80% of the total heat transferred to the risers' panels. For this reason, the radiation enclosure was preferred and was defined in the input deck to account for the radiation heat transfer between the heat structure simulating the reactor vessel (heat structure 101) and the heat structures simulating the risers' front pipes and fins (heat structures 201-9) [12]. Even if

conduction between the fins and pipes was not simulated, the total energy balance within each heat structure would be preserved in the model.

Because each axial node of a given heat structure can be connected to only one volume, the left side of heat structures 201-9 was connected to pipe components 201-9, ignoring the convection heat transfer between the fins and the back cavity. Due to the small dimension of the back cavity and to the expected low temperature of the back side of the pipe walls and fin, this approximation was considered acceptable. Radiation to the back cavity was also omitted. Convection between the pipes' back wall and the back cavity was modeled instead. The heat generation in the reactor vessel (HS 101) was simulated by imposing a constant heat flux as a boundary condition at the left boundary. A convective boundary was defined on the right side of the same heat structure, connected to the single volume simulating the reactor cavity. As also shown in Figure4, a heat structure was included in the model to account for the heat losses by convection to the environment through the cavity walls (HS 115).

Figure 6 helps identify the process followed to define the characteristics of the heat structures simulating the combined risers' front walls and fins.

Figure 6. Risers' Heat Structure Definition.

Cylindrical geometry was specified for the heat structure representing the pipe front walls and fins. Left and right radii were required in the geometry cards of each heat structure. In order to use the view factors as calculated and preserve the reciprocity rules, the right boundary radius was initially calculated in order to satisfy the following condition:

$$2\pi r_{right} L = A_{wall, out} \quad (1)$$

where r_{right} is the equivalent radius of the right boundary surface (in contact with the front cavity, volume 100) and L is the axial length of the wall, $A_{wall, out}$ is the outer surface area of the element, consisting of the front side of the pipe and the two adjacent fin halves. The left radius, r_{left} , was then calculated:

$$r_{left} = r_{right} - t_{wall} \quad (2)$$

In Equation 2, t_{wall} is the thickness of the wall.

Under these conditions, the right boundary surface area of the heat structure reflected the actual surface area of the riser/fin section. This is a required condition for the radiation

enclosure, where the view factors (calculated using the real surface area of the pipes and fins) are defined. On the other hand, the left boundary surface area was overestimated. A correction to the left heat transfer area was made to account for the difference between the area defined in the RELAP5 input deck and the real heat transfer area between pipe half wall and coolant. This was performed using the Fouling factor (F) in the additional left boundary conditions card. This factor is a user-defined convective heat transfer coefficient multiplier. This factor was introduced in order to adjust the total left heat transfer to the proper value:

$$F = \frac{A_{wall,in}}{A_{relap}} \quad (3)$$

where $A_{wall,in}$ is the real inner surface area, and A_{relap} is the left boundary surface area defined in the RELAP5 heat structure geometry cards.

Table 1 lists all the cavity heat structures defined in the reactor cavity model with the heat transfer mechanisms simulated.

Table 1. Cavity Heat Structures Summary

HS #	Component	Convection	Radiation
101	Vessel	✓	✓
115	Cavity Walls	✓	✓
2010-2090	Pipe/Fin Front Walls	✓	✓
2011-2091	Pipes Back Walls	✓	

The radiation view factors required as input parameters for the radiation enclosures defined in the RELAP5-3D input deck were calculated using NEVADA™ software [13], which is based on the Monte Carlo method largely used for radiation analysis. The software allows the user to create the model by drawing the surfaces and imposing the direction of the surface normal vectors. Figure 7 presents the geometry created using Nevada software. All the dimensions and relative radial and vertical positions of vessel, pipes, and fin were taken from the drawing of the facility. Additional walls (not shown in the figure) were included to estimate the view factors between the main structures (vessel and risers' panel) and the cavity walls, ceiling, and floor. As required for any Monte Carlo calculation, the number of "rays" (histories) had to be selected and optimized in order to achieve an acceptable standard deviation despite the constraint on computing time. The number of rays used for the simulation was 10^9 , and the maximum standard deviation achieved in the view factor calculations was 0.07. The emissivity of the surfaces of vessel and pipes defined in the RELAP5 input was assumed equal to 0.8 [14, 15].

Figure 7. NEVADA Input Geometry

The boundary and initial conditions for the simulations (ambient pressure and temperature, initial temperature of the water in the system, tank water level) were assigned the values that were recorded during the experiment.

V. RELAP5-3D MODEL REFINEMENT AND COMPARISON WITH SHAKEDOWN TESTS

Due to the heat losses observed during the facility shakedown, the RELAP5-3D input file needed refinement in order to model the losses through the cavity walls. The original model previously described, already accounted for the heat losses through the pipeline, water tank, and part of the reactor cavity. These heat structures were slightly modified to account for the final configuration of the thermal insulation installed in these regions. The following factors needed to be added:

- The heat losses from the back of the heaters.
- The cavity thermal inertia, in particular for heaters and vessel.

These were incorporated by changing nodalization of the heat structure that simulates the reactor vessel and heaters. Figure 8 shows the comparison between the original nodalization adopted for the cavity heat structure (left) and the new approach proposed (right).

Figure 8. Cavity Heat Structure Nodalization - Original (left), New (right).

In the original nodalization, the reactor vessel and heaters were simulated by a heat structure that had the thickness and the thermal properties (conductivity and heat capacity) of the vessel shell (stainless steel 304). The heaters were not modeled as part of the heat structure. As previously mentioned, a symmetric boundary was imposed in the left side of the heat structure, where a positive heat flux was imposed to simulate the heat from the radiant heaters.

The heat structure radial nodalization was improved in the new model to account for different materials in this region of the cavity (thermal insulation, heaters, reactor vessel shell) and heat losses at the left boundary (heaters' back wall), observed during the experimental shakedown.

To account for the real thermal properties of the layers, three mesh intervals were defined, and thicknesses and materials of vessel, heaters, and thermal insulation were specified in the input file. In particular, stainless steel 304 was used for the reactor vessel, and the heaters were assumed to have similar thermal properties to stainless steel. Thermal properties of the thermal insulation were retrieved from the manufacturer datasheets. A volumetric power source (total power of 6 kW, equal to the one imposed in the experiment at steady-state) was defined in the heaters' mesh interval, and a power table was defined to realistically follow the power ramp used during the experiment. A convective boundary was imposed on both sides of the heat structure, the left side's simulating the heat losses by convection to the environment at the heaters' back walls.

Figure 9 presents the preliminary simulation results obtained with the original model and with the new (modified) RELAP5-3D model, compared to one of the experimental shakedowns

performed prior the final steady-state runs. The results are shown in terms of the volumetric flow rate of the water in the loop during the initial phase of the test, when the power ramp was applied to the heaters.

Figure 9. Shakedown Experimental Data and RELAP5-3D Simulation Results.

The modifications implemented in the model of the reactor cavity produced a better prediction of the transient than did the original RELAP5 model, where the loop water volumetric flow rate was sensibly overestimated.

The simulation results presented in the next section were obtained with the modified model.

VI. RELAP5-3D SIMULATIONS AND COMPARISON WITH THE EXPERIMENTAL DATA

The cavity inlet coolant temperature during the experiment (black thick curve) and the RELAP5-3D predictions (orange curve) are plotted in Figure 10. The uncertainty of the thermocouple measurements is also shown (black thin curves).

Figure 10. Cavity Inlet Coolant Temperature.

The RELAP5-3D predictions are in satisfactory agreement with the experimental data throughout the ramp-up and steady-state phases.

A similar satisfactory agreement with the experimental data was obtained for cavity outlet coolant temperature, shown in Figure 11.

Figure 11. Cavity Outlet Coolant Temperature.

The volumetric flow rate of the main coolant is shown in Figure 3.

Figure 3. Main Coolant Volumetric Flow Rate.

Also in this case, the predictions of the RELAP5-3D model were basically within the uncertainty of the flow measurement during the experiment.

During the initial phase of the experiment, the flow rate sharply decreased and increased during the ramp-up phase. This may be caused by the strong flow asymmetry within the nine risers, due to the asymmetrical geometry of the panel inlet and outlet [6]. In particular, the risers close to the bottom manifold water inlet were characterized by a very low flow, due to the larger pressure losses at the entrance. The flow through these pipes may require a higher temperature

difference to establish natural circulation. Once this temperature difference is reached (likely at approximately 2000 seconds), the coolant started flowing through these risers, producing an overall flow perturbation. This perturbation was also observed in the water outlet temperature once the additional flow from these risers started mixing within the top manifold. The RELAP5-3D simulation does not show such a phenomenon. The authors believe that improvement in the loss coefficients at the risers' inlet and outlet (currently assumed the same for all the risers) may be required to account for more realistic pressure drops.

VII. CONCLUSIONS

A RELAP5-3D input model representing a small-scale RCCS experimental facility was prepared to perform simulation of the system behavior during the startup and steady-state phases of the facility. Special heat structure nodalization techniques were applied to the reactor cavity model, to account for the radiation and convection heat transfer mechanisms in the cavity, thereby maintaining a realistic representation of the complex cavity geometry.

The simulation results showed satisfactory agreement with the experimental data in terms of coolant flow rate and inlet and outlet cavity coolant temperatures, thus confirming the capabilities of RELAP5-3D to predict the behavior of this type of passive system.

After the external heat losses through the cavity walls and other internal structures were included in the RELAP5 model, the model's predictions for the transient phase of the experimental run were consistent with the experimental observation, demonstrating the importance of including specific features of the reactor cavity such as cavity walls, insulation, and other structures to properly account for the cavity thermal inertia.

NOMENCLATURE

A	Heat Transfer Area
F	Fouling Factor
L	Length
r	Radius
t	Thickness

Acronyms

DOE	Department of Energy
HS	Heat Structure
NEUP	Nuclear Energy University Program
RCCS	Reactor Cavity Cooling System
SJ	Single Junction
TV	Time-dependent Volume
VHTR	Very High Temperature Gas-Cooled Reactor

ACKNOWLEDGMENTS

This project was founded by the U.S. Department of Energy and the Nuclear Energy University Program (NEUP).

The authors would like to mention the technical collaboration with the University of Wisconsin, Madison; the University of Idaho; and the Argonne National Laboratory (ANL).

REFERENCES

1. H. D. Gougar, D. A. Petti, R. N. Wright, W. E. Windes, R. R. Schultz, J. S. Herring, P. Sabharwall, P. Humrickhouse, "Current status of VHTR technology development," Proceedings of HTR 2010 Prague, Czech Republic, October 18-20, 2010, Paper 048 (2010).
2. D. Hittner, "RAPHAEL, A European project for the development of HTR/VHTR technology for industrial process heat supply and cogeneration," Proceedings of HTR 2006, 3rd International Topical Meeting on High Temperature Reactor Technology, October 1–4, Johannesburg, South Africa (2006).
3. L. J. Lommers, F. Shahrokhi, J. A. Mayer III, F. H. Southworth, "AREVA HTR concept for near-term deployment," Nuclear Engineering and Design, 251 (October 2012).
4. J. Antwerper, G. P. Greyvenstein, "Evaluation of a detailed radiation heat transfer model in a high temperature reactor systems simulation model," Nuclear Engineering and Design, 238: 2985–2994 (2008).
5. D. A. Dilling, "Passive decay and residual heat removal in the MHTGR," IAEA-TECDOC-757, Juelich (1982).

6. R. Vaghetto, Y. A. Hassan, "Experimental investigation of a scaled water-cooled reactor cavity cooling system," Nuclear Technology (2013).
7. RELAP5-3D Code Manuals, INEEL-EXT-98-00834, Rev. 2.4 (2005).
8. INL/EXT-06-01362, "Development of safety analysis codes and experimental validation for a very high temperature gas-cooled reactor" (2006).
9. R. Vaghetto, W. Huali, Y. A. Hassan, "RELAP5-3D simulations of the reactor cavity cooling system experimental facility," ANS Annual Meeting, Chicago, IL, June 24-28 2012, Paper 979 (2012).
10. R. Vaghetto, H. Wei, Y. A. Hassan, "Simulation of a simple RCCS experiment with RELAP5-3D system code and computational fluid dynamics computer program," 14th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics, NURETH-14, Toronto, Ontario, Canada, September 25-30, Paper 352 (2011).
11. R. Vaghetto, L. Capone, Y. A. Hassan, "Experimental study of the effect of graphite dispersion on the heat transfer phenomena in a reactor cavity cooling system (RCCS)," Nuclear Technology, 177(2) (2012).
12. D. D. Lisowski, T. C. Haskin, A. Tokuhiko, M. Anderson, M. L. Corradini, "Study of the behavior of an asymmetrically heated reactor cavity cooling system with water in single phase," Nuclear Technology, 183 (2013).
13. TAC Technologies, "NEVADA Software Package Quick Reference Series," TAC Technologies, USA (2000).
14. C. H. Oh, G. C. Park, C. Davis, "RCCS experiments and validation for high-temperature gas-cooled reactor," Nuclear Technology, 167 (2009).

15. A. Sala, "Radiant Properties of Materials," Elsevier Science Publishing Co., Inc., New York (1986).