ANALYTICAL MODELING OF A SCALED REACTOR CAVITY COOLING SYSTEM (RCCS) WITH AIR

Y. Aharon[1,2], M. Muci[1], C.Tompkins[1], M.Anderson[1], M. Corradini[1]

[1]: Department of Engineering Physics, University of Wisconsin-Madison, 1500 Engineering Drive, Madison, WI 53706, USA

[2]: NRCN, P.O.Box 9001 Beer Sheva, Israel

Email: aharon@wisc.edu

ABSTRACT

The present work describes an analytical modeling of experiments that were conducted in a scaled air-cooled Reactor Cavity Cooling System (RCCS). This quarter-scale RCCS facility was designed and built at UW-Madison to simulate the full-scale air-cooled (General Atomic) RCCS design concept for passive cooling of a Modular High Temperature Gas Reactor (MHTGR) pressure vessel. The experimental study was conducted to investigate the thermal hydraulic behavior and the heat removal performance of the RCCS with air. That system consists of vertical parallel riser tubes located along the reactor cavity wall and open to the atmosphere. The heat from reactor pressure vessel is radiated to the air-cooled riser tubes and the air flows by natural circulation with heat air discharged into the atmosphere. The quarter-scale RCCS was run on three modes: Forced flow testing, natural circulation in constant heat flux and natural circulation with an asymmetric heat flux. The experiments were conducted on various heating power 9.91-37.97 kW, which span the full-scale range of heating corresponding to decay heat.

The model in the present work is based on elementary one dimension radiation heat transfer equation and convection correlation from the literature. The heater and the risers were divided into axial elements and heat balance equation was written for each element. A good agreement was achieved between the experiments risers walls and air temperatures measurement and the model prediction. For the natural circulation experiments, good agreement with the airflow rate was achieved within 9% of the measured value.

KEYWORDS: RCCS, natural circulation, MHTGR, riser

1. INTRODUCTION

Passive reactor cooling systems became more attractive over the last two decades. This subject was studied in many published papers, which deal with reactor safety. Passive cooling systems, when compared to active systems, do not depend on an external energy sources (AC power) and uses natural phenomena, such as gravity, conduction and radiation, which are always present.

That concept is used in existing reactors for certain safety systems and is the basis for future reactors design such as the Generation IV reactors. In some of the reactors the passive cooling system is designed to remove the decay heat from the core by natural circulation, which flow through the core after an accident such as loss-of-coolant-accident (LOCA). The coolant (gas or water) circulated through the core by the buoyancy mechanism and then flows through a heat exchanger, which cooled by another coolant (air or water) by natural or active means. That concept was studied by F. J. Mackay et al.[1] and by Michael A. P. et al.[2] for the Gas Cooled Fast Reactor (GFR) in various accident scenarios. Some of the reactors such as the IRIS (K. Shirvana,[3]) are based on the natural circulation cooling in a nominal conditions to increase the reliability of reactor safety systems. Another type of natural circulation reactor cooling system is the Reactor Cavity Cooling System (RCCS), which cools the reactor cavity, and in case of an accident, removes the decay heat which transferred from the reactor vessel wall. C. Oh et al.[4] summarized the different RCCS types which are under development in the HTGR. Some of them are based on a water forced or natural circulation and the other are based on an air natural circulation cooling.

The General Atomics (GA) Modular High Temperature Gas Reactor (MHTGR) is a modular reactor design with a capacity of 450 MWt per module. It has a primary helium coolant loop and graphite moderator. The use of helium as the coolant in combination with a graphite moderator offers enhanced neutronic and thermal efficiencies and makes possible production of high temperature nuclear heat, and hence the name, High Temperature Gas-cooled Reactor (HTGR). For that reason that reactor type is one of the Next Genaration Nuclear Plant which is under research in those days. One of the safety issue of that reactor which is in research is the decay heat removing in case of accident. One of the means for removing decay heat is provided by a safety-related Reactor Cavity Cooling System (RCCS). A schematic of the RCCS is shown in Figure 1. The RCCS removes heat radiated from the un-insulated reactor vessel by natural circulation of outside air through enclosed cooling panels along the reactor cavity walls. Because air naturally circulates through the RCCS continuously, it is always available to remove decay heat under accident conditions without reliance on active components, power supplies, or operator action. The heat sink for that system is the surrounding air, which provides unlimited working time. The RCCS provides also cooling of the reactor cavity concrete during normal operation. The RCCS offers a clear advantage compared to forced cooling systems in that it does not require electrical power and can in theory, operate indefinitely in an accident scenario.

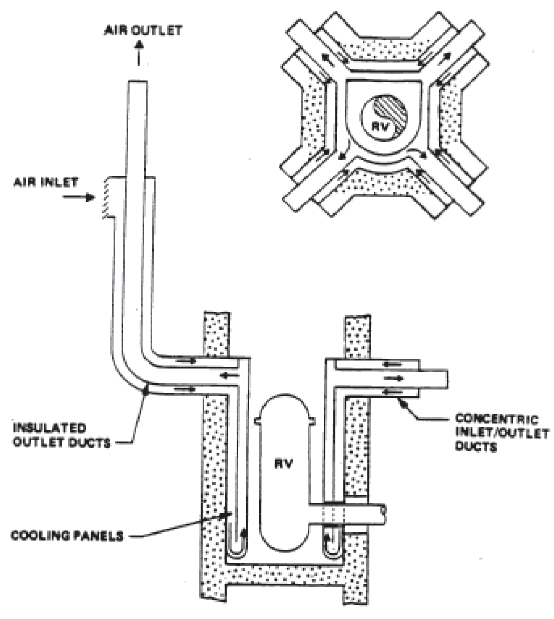

Figure 1. Reactor cavity cooling system conceptual schematic.

In the GA MHTGR design the RCCS is a secondary cooling loop in which 227 rectangular ducts (5 cm by 24 cm) line the concrete containment around the reactor pressure vessel. The rectangular ducts are connected to two sets of four chimneys. In each of the two sets, there are two alternating groups of a hot outlet chimney and a cold inlet chimney.

The cooling performance of the RCCS was studied in two law scale experimental systems. One is the Natural convection Shutdown heat removal Test Facility (NSTF) which is a half-scale air cooled RCCS in the Argonne National Laboratory (ANL) and the second one is quarter-scale air cooled RCCS in UW-Madison. Both of the experimental systems are based on the General Atomics RCCS design concept (Lomperski,[5]). An intensive experimental research was conducted by M. A. Muci [6] to evaluate the heat removal performance at steady-state conditions of the RCCS. Those experiments were conducted in a forced and natural circulation conditions, while in the natural circulation condition a uniform and asymmetric heat flux conditions were examined.

This present work presents a prediction of the experimental results by using an analytical model.

2. THE EXPERIMENTAL SYSTEM AND EXPERIMENTS MATRIX

The details of the experimental phase of this research are presented in ref. [6]. The cavity plane view of the designed GA MHTGR reactor vessel and RCCS, is presented in figure 2.

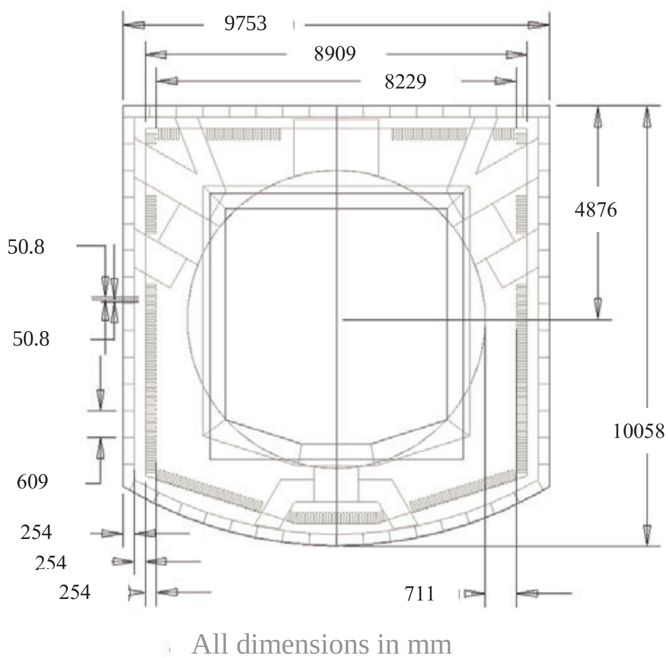

Figure 2. Cavity plane view of reactor vessel and RCCS[6]

The scaled air RCCS facility at UW-Madison is a quarter-scale reduced length experiment housing six riser ducts that represent a 9.5° sector slice of the full-scale GA air RCCS concept. The air RCCS facility consists of three important components: inlet plenum, heated cavity, and the outlet plenum/exhaust ducts. The inlet plenum is the entry point for air drawn from the environment by the air RCCS. Electrical resistance heaters inside the heated cavity simulate the reactor pressure vessel of the reactor and radiate heat to the six riser ducts. The outlet plenum provides a volume to allow mixing before the heated air returns to the outside environment via two exhaust ducts. A schematic view of the air RCCS facility components presents in figure 3.

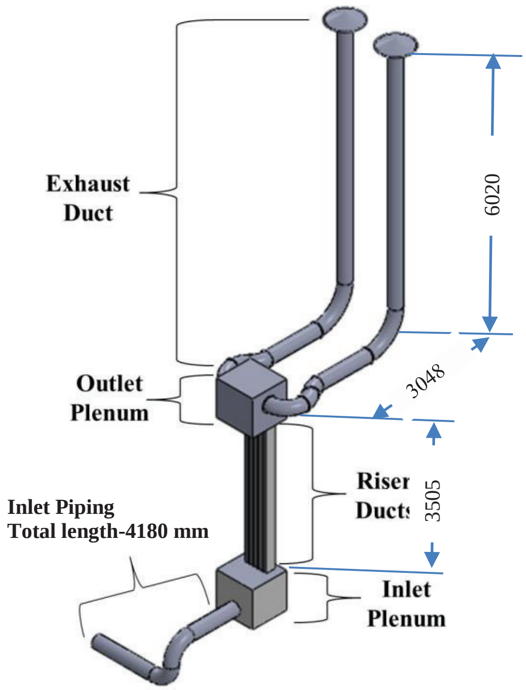

Figure 3. Schematic view of the air RCCS facility components [6] (in mm)

The six riser ducts (rectangular ducts) in the quarter-scale facility have the same dimension and spacing as the full-scale GA MHTGR reactor cavity cooling system design. The risers are 25.4 mm (width) by 254 mm (length) and 25.4 mm of spacing was placed in between risers. The risers height is 3759 mm and the total high of the experimental flow path from the inlet piping to the exit of the exhaust duct is 13 meters. The six risers are located inside the heated cavity, whose main function is to serve as a thermal enclosure for the heat transfer from the radiant heaters to the six riser ducts and to minimize heat losses to the outside environment. Natural convection cells develop inside the heated cavity much like they would in the reactor cavity of the GA MHTGR. A plane view of the heated cavity is presented in figure 4. As it can be seen in the figure, four rows of electrical heaters were mounted on one end of the heated cavity in the front of the narrow side edge of the risers. Instrumentation was placed in the air RCCS facility to control the heating zones and to record temperature and velocity measurements at certain locations. Flow velocity was measured at the inlet of the inlet piping and at the inlet of the risers. Air temperature was measured at the inlet of the inlet piping, at the inlet of each of the exhoust ducts and inside the risers. The vertical location of temperature measurements along the risers is presented in figure 4. It can be seen that riser #4 was more heavily instrumented. In that riser the edge effect is smaller and the measured data may simulate better the full scale case. In risers #1-3 and 5-6 the thermocouples located to measure the air temperature inside the riser and the front side edge of the riser at three levels along the riser as presented in figure 5. In riser #4, the temperature measurement was conducted in seven levels along the riser. In each measurement location in this riser thermocouples were connected to the front, side (left and right) and backside edges of the riser wall except of the air temperature measurement.

The reported[6] uncertainties of the temperatures measurements was ±0.7 ^{o}C for all the temperatures measurements except of the front riser's temperature uncertainty, which was ±2.2 ^{o}C. For the velocity measurement at the inlet piping, the reported manufacturer uncertainty as ±2.0% of the reading.

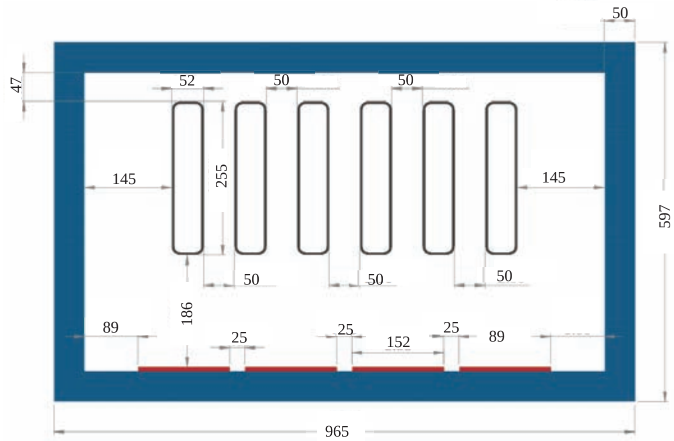

Figure 4. Plane view and dimensions (in mm) of the experimental heated cavity [6]

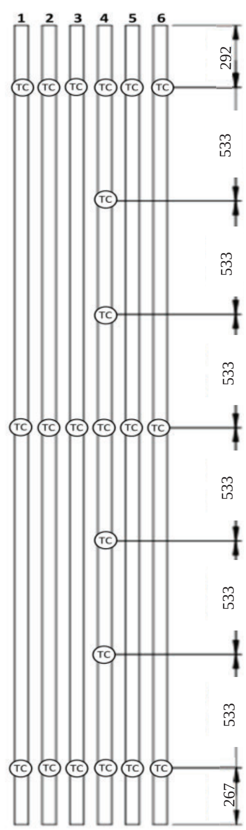

Figure 5. Riser surface thermocouple locations (in mm) [6]

Three various types of experiments were conducted with the experimental system. The first type was a forced flow test at two heating powers of 19.82 kW and 37.97 kW. In this experiment type a blower was used to circulate the air through the system. The second type was a natural circulation test, which was conducted at heating powers of 19.82 and 37.97 kW. The third type is a natural circulation with asymmetric heat flux in which just two rows of heater were operated in the heated cavity.

In this present work, a modeling of the first and the second experiments type is presented.

3. THE THERMAL MODEL

3.1. The Heating Cavity Zone

The heat transfer from the heater in the experiment system to the risers and the heating cavity walls by natural convection and radiation. Figure 6 presents the main details of the model. For modeling of the experimental system a one-dimensional heat conduction, radiation and flow assumption was used. In the model, the risers as well as the heaters were divided into 30 axial segments as it presented in figure 6 (a).

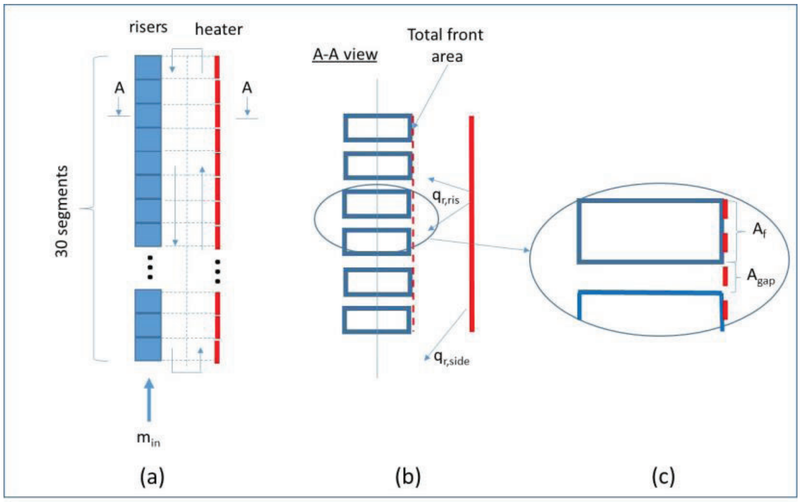

Figure 6. Thermal modeling of the experimental system

It was assumed that the heater is a homogenous plane heat source with heat capacity of ceramic material. The heater exchange heat by radiation with two different surfaces. One is the front surface of the risers ("$q_{r,ris}$" and "total front area" in figure 6 (b)) and the second is the side wall ($q_{r,side}$) of the heating cavity. The radiation heat flux which reaching those surfaces from the heater is calculated as follows:

$$q_i^{"}(j) = F_{H,i}\sigma\varepsilon_i(T_H{}^4(j) - T_i^4(j)) \tag{1}$$

In this equation the subscripts H and i present the heater and surface respectively. The index j presents the calculated segment along the heater and the risers. As it can be understood from equation (1) that it was assumed (for the simplicity of the model) that each segment of the heater radiate heat only to the segment of the riser with the same index.

The view factor between the heater and the other surfaces was calculated as radiation between two parallel rectangular strips according to the geometry of the system. The radiation which reaches the "total front area", divided to two parts. One is the radiation which heats the front side edge of the riser and the second part heats the side edge of the risers (A_f and A_{gap} in figure 6 (c) respectively). The view factor between the heater and each part was proportional to the ratio between the two different zones. It was assumed that each segment of the heater "see" just the segment with the same index in front of him and the side wall of the heated cavity separately (radiation heat transfer between two bodies). The energy which heating each segment of the risers wall (front or side edge) was calculated by multiplying the heat flux from equation (1) by the calculated segment's area.

The heat exchange with the side wall of the heating cavity ($q_{r,side}$ in figure 6(b)) is based on the view factor between the heater and the side wall. The side wall's temperature which used in the radiation calculation was the average measured value from the experimental results. Actually, that temperature can be calculated based on the heat balance and the insulation properties of the side wall, but due to the uncertainties on the effective properties of those walls (construction effects such as contact resistance thermal bridges etc.), the measured wall temperature was used in the model. Because of the assumption in the model that there is no heat exchange between the side wall and the risers, the heat which reaches the side wall, transfers out of the system and it is the only zone which take into account heat losses from the system. The heat losses in this case can be calculated by summation of the heat transferred from each heater segment to the side walls. In the experiments the heat losses were calculated as the difference between the electrical heating power and the heat removed by the flowing air through the risers.

The second heat transfer mechanism between the heater and the risers is natural convection. For modeling that mechanism, the volume of the heating cavity between the heaters and the risers divided into a 60 cells, which act as a control volume as it is presented in figure 6 (a). One dimension flow was assumed near the heater wall through the adjacent cells in the up direction and in the opposite direction through the cells near the risers. In this model it was assumed that there is now mixing between the streams in the both direction and the natural circulation inside the cavity creating a single flow loop. The convective heat transfer coefficient inside the cavity from the heater and to the risers was calculated based on the local Rayleigh number and using the following correlation [7] for natural convection from vertical wall with constant heat flux.

$$\begin{aligned} Nu_x &= \frac{hx}{k_{air}} = 0.6Ra_x^{0.2} \qquad \text{for Ra} < 10^{13} \\ \text{Nu}_x &= 0.17Ra^{0.25} \qquad \text{for Re} > 10^{13} \end{aligned} \tag{2}$$

The heat flux in this correlation was the heater nominal heat flux and it was used for the convection calculation along the heater as well as the risers zone. The coordinate x was measured from the bottom edge of the heater and from the top edge of the risers for the heater wall and the risers calculations, respectively. The convective heat transfer coefficient at the risers zone was used to calculate the heat transfer from the air to each wall (front, sides and back) separately. An energy balance was used to calculate the heaters and the risers walls segments temperatures as well as the temperature at each cell. For the heater segment and the risers walls segments the following equation was used:

$$\frac{dT_{wall}(j)}{dt} = \frac{\sum q_{conv} + \sum q_{rad} + \sum q_{cond}}{M_{wall}(j)Cp_{wall}} \tag{3}$$

In equation (3), a transient calculation is presented for the walls segment (j) temperature. q_{conv}, q_{rad} and q_{cond} are the amount of energy that transferred to/from the segment by convection, radiation and conduction, respectively. M_{wall} and Cp_{wall} are the segment mass and heat capacity respectively. For the air in the cavity equation 4 is used:

$$\frac{dT_{air}(j)}{dt} = \frac{q_{conv} + \dot{m}_{air}\text{Cp}(\text{T}_{\text{in}}(j) - T_{out}(j))}{M(j)Cp} \tag{4}$$

In equation (4), $T_{in}(j)$ and $T_{out}(j)$ are the inlet and outlet air temperatures to cell j respectively. M is the air mass inside the cell j and q_{conv} is the amount of heat that transfer from the heater wall to the air or from the air to the riser wall. The mass flow rate was calculated iteratively and the converging condition was that the inlet air temperature to the firs cell in the heater side is the exit air temperature of the last (bottom) cell in the riser's side.

3.2 The Riser Zone

For modeling the riser segments, each segment divided to three zones: the front wall, the side wall and the back wall. Each of that zones assumed as a separate body which heated by convection and radiation from the heaters and transfers heat with the adjacent body by conduction and with all the bodies of the segment by radiation inside the segment. The details of this part of the model is presented in figure 7. The view factor between the segment side edges was calculated according to the segment inside geometry and it was assumed that the front side of the segment transfers heat by radiation with the side and the back walls separately and now radiation heat transfer between the side and the back wall.

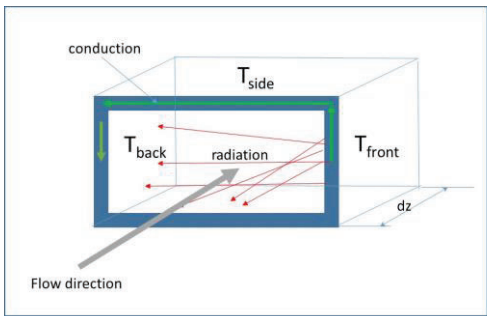

Figure 7. Thermal modeling of riser segment

The conduction heat transfer between the walls of segment (j) was calculated as follows:

$$q_{cond}(j) = k_{steel} \frac{tdz}{l_i}(T_{front}(j) - T_{side}(j)) \tag{5}$$

In equation (5), k_{ss} is the risers wall thermal conductivity, t is the walls thickness, dz is the segments length and the length l_i is the distance between the center of each wall and the adjacent wall along the channel perimeter.

Inside the riser except of the heat transfer mechanisms that described above the air is flowing by forced or natural circulation and removing the heat from the walls. The air flow causing a convection heat transfer and the convection heat transfer coefficient was calculated based on the Reynolds number inside the channel as follows:

$$\begin{aligned} \mathrm{Nu}_{\infty} &= 5 && \text{for Re} < 2300 \\ \mathrm{Nu}_{\infty} &= 0.023\mathrm{Re}^{0.8}\,\mathrm{Pr}^{0.4} && \text{for Re} > 2300 \end{aligned} \tag{6}$$

In this equation Nu_∞ is Nusselt number for developed flow conditions. In the inlet zone of the riser the flow is developing and the local Nu number was calculated based on experimental results of J. Park et al. [8] which fitted to the follow function. The hydraulic diameter (D_h) in those experiments was 51 mm.

$$\mathrm{Nu}(\mathrm{j}) = \mathrm{Nu}_\infty \left(1 + \frac{1.5}{\left(jdz/D_h \right)^{1.25}} \right) \tag{7}$$

The local convective heat transfer coefficient was calculated from the local Nusselt number (eq. 7) and it was applied for the heat transfer from each wall of the segment to the bulk local air temperature. The local air bulk temperature inside each segment inside the riser was calculated in the same way as the air temperature in the heated cavity (equation 4):

$$\frac{dT_{air,ris}(j)}{dt} = \frac{\sum_i q_{conv,i} + \dot{m}_{air} \mathrm{Cp}\left(\mathrm{T}_{\mathrm{in,ris}}(j) - T_{out,ris}(j) \right)}{M_{ris}(j) Cp} \tag{8}$$

In equation (8) $T_{in,ris}(j)$ and $T_{out,ris}(j)$ are the inlet and outlet air temperatures to cell j inside the riser, respectively. M_{ris} is the air mass inside the cell j and $q_{conv,i}$ is the amount of heat that transfer from the riser's inside wall (i) to the air. The mass flow rate is the nominal measured value in case of forced circulation while in case of natural circulation that value was calculated iteratively for matching the momentum balance as will be described in the next paragraphs. It was assumed in this model that the flow rate is evenly distributed through the 6 risers. The flow rate measurement at the inlet of the, risers which was conducted in one of the natural circulating tests, shows that the variation between the risers is less that 1%.

3.3 Momentum Balance

The heat sink of the experimental system is the air flow inside the risers. The present model simulates two types of experiments: the forced circulation experiments and the natural circulation experiments. In the forced circulation experiments for a single riser, the mass flow rate which was used is the total measured value divided by the number of the risers. In the natural circulation model a flow rate was assumed and after reaching a steady state condition, the buoyancy moving force and total pressure losses were calculated. The final mass flow rate was the value in which the buoyancy moving force was equal to the total pressure losses.

The buoyancy moving force was calculated as follow:

$$\Delta p_{boy} = \sum_j g\left(\rho_{in} - \rho(j) \right) \Delta z \tag{9}$$

While ρ_{in} and $\rho(j)$ are the inlet (ambient) and local air density and Δz is a segment length in the vertical direction. In this equation the local air density was calculated based on the local air temperature inside the riser. At the inlet zone of the system (inlet piping) the densities difference will be zero because the air is at the ambient temperature. At the exhaust duct, the densities difference will be a constant value where $\rho(j)$ is referring to the risers exit temperature (neglecting heat losses to the surrounding). The total pressure losses were calculated based on the following equation:

$$\Delta p_{loss} = \sum_j \left(f \frac{L}{D_h} \rho \frac{\mathrm{v}^2}{2} \right)_j + \sum_k \left(K \rho \frac{\mathrm{v}^2}{2} \right)_k \tag{10}$$

In this equation, the velocity v was calculated based on the specific mass flow rate at the channel and its geometry. The friction coefficient f was calculated based on the channel Reynolds number by assuming a smooth channel. The local pressure losses coefficient K was taken from an engineering tables according to the geometry.

As it was presented in the previous equations the calculating mode of the model was a transient calculation while the initial conditions where uniform temperature at all of the system calculating zones (solid and air). The experimental heating power was applied in the model and the measured mass flow rate in the case of forced circulating experiments. In the natural circulation model, the mass flow rate was varied for each calculation until reaching equality between the buoyancy moving force and the total pressure losses. Since the pressure drop measurement in the natural circulation mode is very complicated, the only value for that parameter is calculated based on equations (9) or (10) as it will be presented later. The inlet temperature to the riser was the measured ambient temperature and since there was no measured value of the risers wall emissivity, the value which was taken to of all the radiated surfaces was about unity. That value based on the riser's walls material and color. Anyway to get the heater measured temperature, the heater emissivity value was changed until a match was achieved. The other calculated values and temperatures are not sensitive to the variation of the heater emissivity.

4. RESULTS AND DISCUSSION

The model that presented in the previous section was used for simulation four experiments which reported in reference 6. Two experiments were conducted in forced circulation mode in heating power of 19.82 kW and 37.97 kW and two were conducted in natural circulation mode at the same heating power. The required heating power in the experiments was calculated based on the power scaling of the GA air RCCS design and the decay heat in case of an accident where active cooling systems are incapacitated. Figure 8 presents a comparison between the calculated and measured values in the forced circulating test with heating power of 19.82 kW. The comparison is between temperatures of the air along riser #4 and the riser's wall temperatures. As it was explained, the wall temperature was measured on the front of the riser (the side that facing the heater) on the side and on the back wall of the riser. As it can be seen, the agreement the air measured and calculated temperatures is excellent. The comparison between the walls measured and calculated value is also good. The heat losses in this experiment were calculated by summation of the heat transferred between each heater segment and the side wall. It was found that the heat losses are 3.2 kW which are about 16% of the nominal heater power (4% lower than the reported value in reference 6). In figure 9 the same comparison is presented for the higher power of 37.97 kW in forced circulation test and also hear the agreement between the experimental and the predicted values of the model is good.

Figures 10 and 11 present a comparison between the calculated and measured values in the natural circulating test with heating power of 19.82 kW and 37.97 kW respectively. Also hear a good agreement was achieved in the higher heating power of 37.97 kW. The measured mass flow rate was 0.18 kg/sec while the calculated value is 0.164 kg/sec, which is 9% lower than the measured value. In this experiment the total calculated pressure drop along the system is about 23.5 N/m^2. In modeling of the lower power natural circulating test, under prediction of the risers temperatures was achieved. Figure 11 presents the comparison after reducing the convective heat transfer coefficient inside the risers by 40%. The agreement between the calculated and measured mass flow rate is also very good.

The measured mass flow rate value is 0.15 kg/sec while the calculated value (before correction of the convection heat transfer coefficient) is 0.143 kg/sec which is about 5% lower than the measured value. After the correction the under prediction increased to 7% which is also a very good result. The total pressure drop along the system in this heating power is about 15.1 N/m^2.

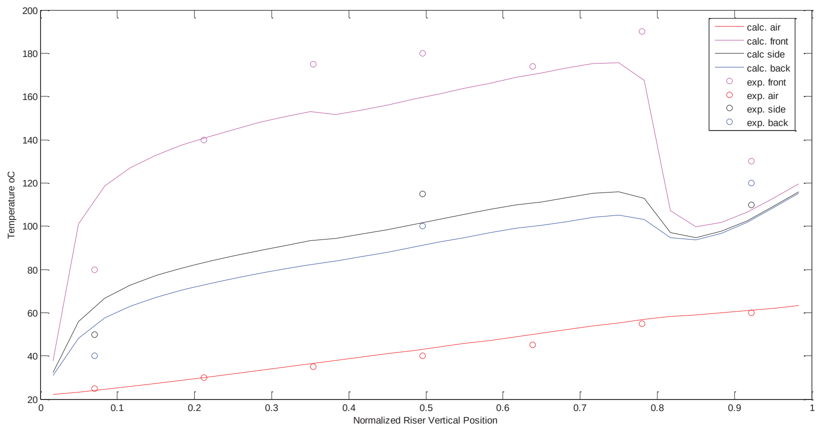

Figure 8. Comparison between calculated and experimental temperatures measurements in forced circulation test (heating power 19.82 kW)

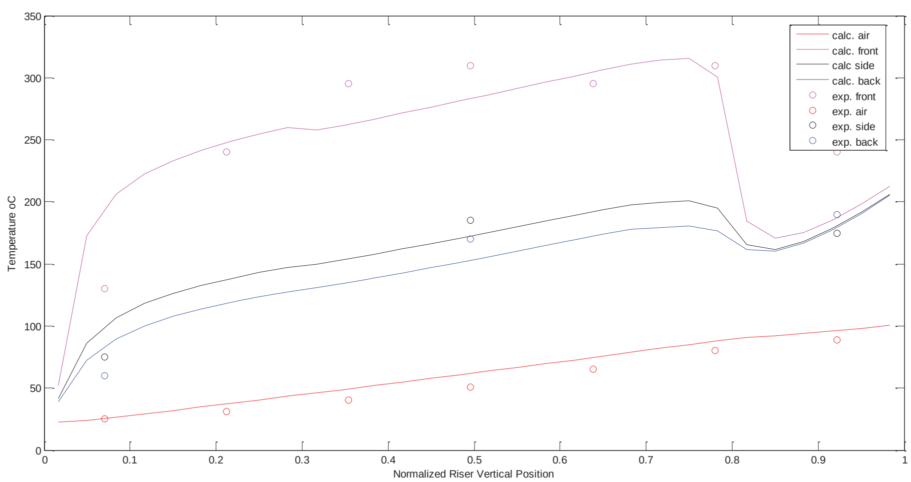

Figure 9. Comparison between calculated and experimental temperatures measurements in forced circulation test (heating power 37.97 kW)

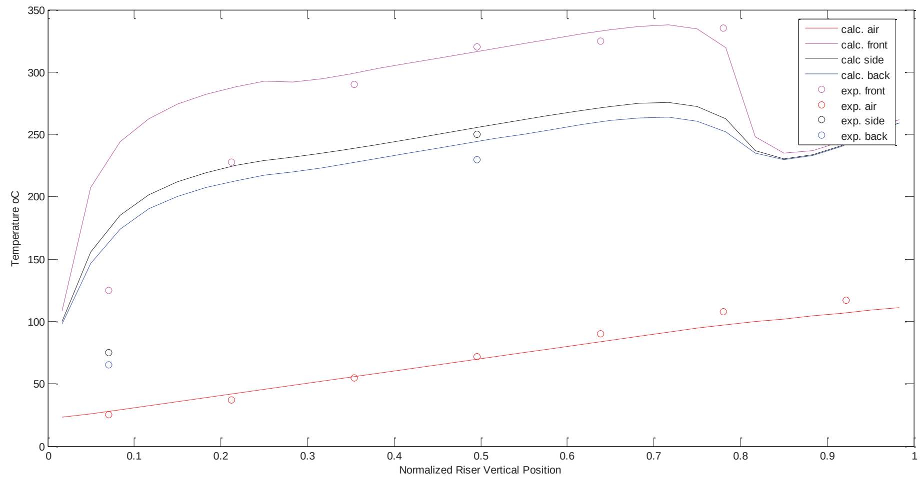

Figure 10. Comparison between calculated and experimental temperatures measurements in natural circulation test (heating power 19.82 kW)

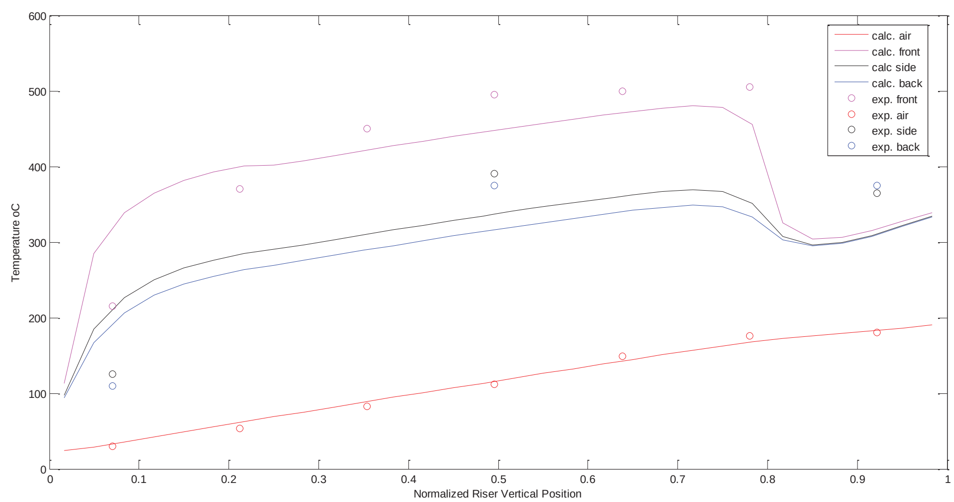

Figure 11. Comparison between calculated and experimental temperatures measurements in natural circulation test (heating power 37.97 kW)

5. SUMMARY AND CONCLUSIONS

In the present work an analytical model was presented for calculation of experimental results which conducted with the quarter-scale air cooled RCCS in UW-Madison. In the model the experimental system which includes the heater, the heating cavity and the risers divided into one dimensional segments and cells and a the energy conservation equation was solved for each segment / cell for calculation of the transient behavior of the various elements temperatures. The heat transfer mechanisms, which took into account in the model where convection and radiation between the heaters and the risers walls, conduction and radiation between the risers inside walls and convection between the risers inside walls and the air which flow inside the risers and acts as a heat sink. The calculation of the convection heat transfer coefficient inside the heating cavity and inside the risers based on correlation from the literature. Two types of experiments were calculated with the model, forced circulating test and one natural circulating test. In the forced circulation test the nominal measured air mass flow rate was used in the model. Comparison between the measured temperatures of the air inside the risers and the risers walls show a good agreement.

In the natural circulation model, the air mass flow rate was also calculated, based on a momentum equation and balance between the buoyancy moving force and the total pressure losses. In the two experiments, which simulated in the model, the disagreement between the predicted and the measured mass flow rate was no bigger than 9%. In this model a good agreement was achieved between the measured and calculated air and risers temperatures in the high heating power. In the lower heating power the model underpredicts the experimental temperature and reducing of the convection heat transfer coefficient inside the risers by 40% was needed for getting a reasonable agreement. The calculated heat losses value was about 16 % of the nominal heater power. That value is lower than the reported heat losses by 4%.

The developed model limited to the experimental geometry, which was used in the experiments, but it can be change to another riser's geometry. Anyway, for more complicate surfaces geometries the user of the model will need few more simplifications and assumptions in order to use this model. The model is also need more validation work on the natural circulation version.

REFFERENCES

1. F. J. Mackay, G. E. Apostolakis, P. Hejzlar, "Incorporating Relability into the Design of Passive Cooling System with Application to Gas-Cooled Reactor", Nucl. Eng. Design, **238**, pp. 217-228, (2008)
2. M. A. Pope, J. I. Lee, P. Hejzlar and M. J. Driscoll, "Thermal Hydraulic Challenges of Gas Cooled Fast Reactors with Passive Safety Features", Nucl. Eng. Design, **239**, pp. 840-854, (2009)
3. K. Shirvan, P. Hejzlar and M. S. Kazimi, "The Design of a Compact Integral Medium Size PWR", Nucl. Eng. Design, **243**, pp. 393-403, (2012)
4. C. Oh, C. Davis and G. C. Park,"RCCS Experiments and Validation for High Temperature Gas Cooled Reactors", proceedings of NURETH – 12, log Number 180, (2007)
5. S. Lomperski, W.D. Pointer, C. P. Tzanos, T.Y.C. Wei, A.R. Kraus, "Air-Cooled Option RCCS Studies and NSTF Preparation", ANL-GenIV-179, Argonne National Laboratory, (2011)
6. M.A. Muci, "Thermal Hydraulic Analysis of an Experimental Reactor Cavity Cooling System with Air", M.Sc thesis University of Wisconsin-Medison, 2014
7. J. P. Holman, "*Heat Transfer*", 10th Edition, pp. 336-337, McGraw-Hill, New York (2010)
8. J. S. Park, J. C. Han, Y. Huang, S. Ou and R. J. Boyle, "Heat Transfer Performance Comparisons of Five Different Rectangular Channels with Parallel Angled Ribs", Int. J. Heat Mass Transfer, **35** (11), pp. 2891-2903, (1992)