NURETH14-352

SIMULATION OF A SIMPLE RCCS EXPERIMENT WITH RELAP5-3D SYSTEM CODE AND COMPUTATIONAL FLUID DYNAMICS COMPUTER PROGRAM

R.Vaghetto[1], H. Wei[1] and Y.A.Hassan[1]
[1] Texas A&M University, Texas, USA

Abstract

A small scale experimental facility was designed to study the thermal hydraulic phenomena in the Reactor Cavity Cooling System (RCCS). The facility was scaled down from the full scale RCCS system by applying scaling laws. A set of RELAP5-3D simulations were performed to confirm the scaling calculations, and to refine and optimize the facility's configuration, instrumentation selection, and layout. Computational Fluid Dynamics (CFD) calculations using StarCCM+ were performed in order to study the flow patterns and two-phase water behavior in selected locations of the facility where expected complex flow structure occurs.

Introduction

The Very High Temperature Reactor (VHTR) was identified to be one the reactor types of the Next Generation Nuclear Plant (NGNP). Due to its high operating temperatures, new safety systems will be included in the reactor design. The Reactor Cavity Cooling System (RCCS) is one of these safety systems designed to guarantee the integrity of the fuel, the reactor vessel and the structures inside the reactor cavity by removing the heat during both normal operation and accident scenarios. The designed experimental facility is a small scale water-cooled model option which is aimed to study the complex thermohydraulic phenomena in the RCCS during an accident scenario. Appropriate scaling laws were applied to calculate the main dimensions of the facility from the full scale plant. A first set of steady-state simulations was performed using RELAP5-3D system computer program [1] Due to the complexity of the expected phenomena of the system, RELAP5-3D was also used to predict the main thermohydraulic parameters, such as fluid and surface wall temperatures, flow regimes at different locations of the facility and the overall system time response. . These predictions are used to delineate the phenomena that will be observed during the experimental performance, and to refine, optimize the facility's configuration and instrumentation layout. As it will be shown in the next section, the results obtained were also used to implement some safety precautions in the experimental facility to avoid damage of facility components. A sensitivity analysis was also performed to assess the capability of the code of prediction of some of the phenomena of the RCCS such as coolant recirculation between adjacent risers. CFD calculations were also performed in order to study the behaviour of the flow in selected locations where the expected three dimensional behaviour. This would overcome some limitation of system codes.

1. Full plant and experimental facility overview

A typical RCCS layout of the VHTR is presented in Figure. 1. The reactor vessel is placed inside the reactor cavity and fixed to the concrete walls by stainless steel supports.

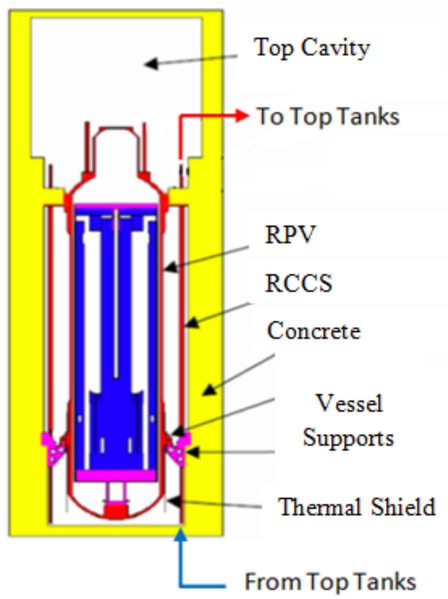


Figure 1. Reactor Cavity Cooling System Layout

The Reactor Pressurized Vessel (RPV) is surrounded by stainless steel vertical pipes (risers) which cover the surface of the reactor cavity. Risers are organized in panels. The reactor configuration considered in this paperwork consists of 25 riser panels each containing 9 risers. Eight water storage tanks (indicated but not shown in the figure) are placed on top of the cavity, collecting water from riser panels. Cold water coming for the tanks moves downward through the downcomers and collects in cold manifolds from which is then distributed to the risers. The buoyancy forces induced by the coolant density gradient insider the risers will drive the coolant flow which now moves upward through the risers, collects into the top hot manifold and reaches the water tanks. During off-normal system operations, the ultimate heat sink is the environment where the steam produced by the evaporation of the water in the tanks is discharged. The RCCS is currently being designed to remove 0.7MW during normal operation and 1.5MW in case of accident scenario. Among the heat transfer mechanisms taking place inside the cavity, the radiation has been confirmed to be one of the most important mechanisms by which the heat is transferred to the RCCS [2]. Different proposals were considered in order to enhance the heat transfer by radiation including thin metal panels welded on the back of the pipes (shield configuration) or along the midplane of the pipes (fin configuration). The small scale experimental facility consists of 9 risers, two manifolds (top and bottom) and one cylindrical water tank. An overview of the facility and the cavity is presented in Figure 2.

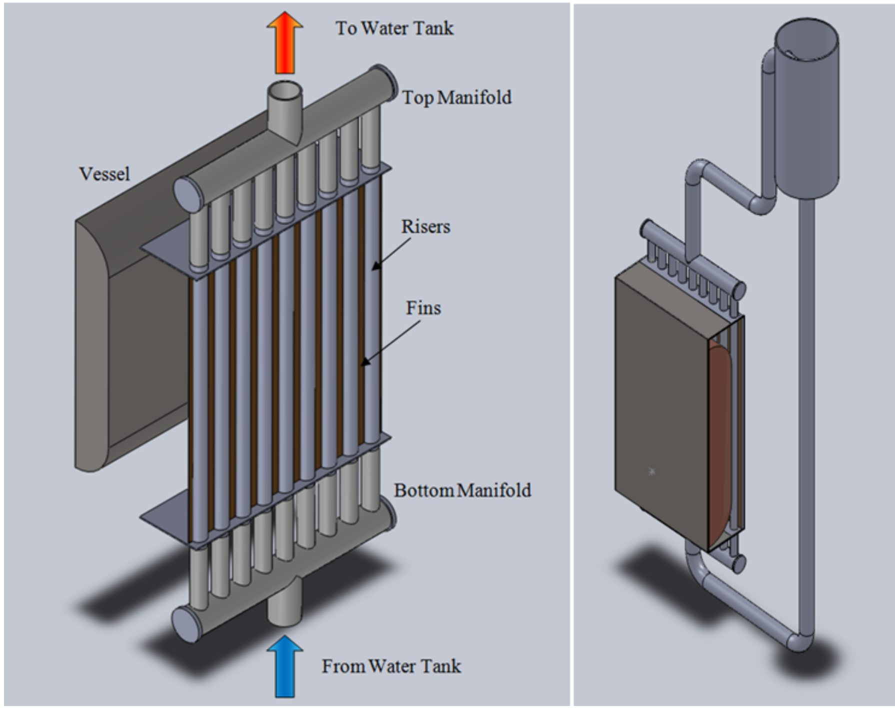


Figure 2. Experimental Facility Overview

Two manifolds (hot manifold at the top and cold manifold at the bottom), the reactor vessel in front of the risers and the water tank on top of the cavity are shown in Fig. 2. The risers are connected via a thin metal panel welded along the midplane (fin configuration). The facility represents only one portion of the cavity, modelling one panel and the corresponding surface of the reactor vessel (approximately 14°). This feature allows adopting a planar geometry for the vessel and the cavity.

2. Facility scaling

The down-scaling of the facility was performed with the guidance of the activity carried out on similar facilities [3]. One of the main dimensions of the plant that was scaled was the axial length (including reactor vessel and cavity). The similarity length l_R, defined as the ratio between the characteristic length of the model and the characteristic length of the prototype, was calculated assuming the axial length of the reactor vessel in the experimental facility to be equal to 1m, so that:

$$l_R = \frac{l_{model}}{l_{prototype}} = \frac{1m}{21.84m} = 0.046 \tag{1}$$

Due to the predominant importance of the radiation heat transfer through the cavity [2], the same similarity length was applied to scale the radial dimensions of the facility. The radial view factors of the full plant and the scaled facility were estimated to confirm that the change in the radial view factor induced by the scaling was negligible. Additional parameters of interest were used during the scaling phase, such as the Froude number, ratio of the inertial forces and gravity forces, and the Richardson number, which expresses the ratio of potential to kinetic energy:

$$Fr = \frac{U_o^2}{gl} \tag{2}$$

$$Ri = \frac{g\beta(T_{hot} - T_{ref})l}{U_o^2} \tag{3}$$

A unity similarity between the model and the prototype was assumed for both Froude and Richardson numbers ($Fr_{model}/Fr_{prototype}=1$, $Ri_{model}/Ri_{prototype}=1$). In addition, the temperatures between the model and prototype were assumed to be preserved and, subsequently, all the properties of the coolant such as density and specific heat. Table 1 summarizes the scaling ratios that were directly calculated from the considerations previously mentioned.

Scaling Ratio	Description	Value
U_{oR}	Scaled Reference Velocity	$l_R^{0.5}$
t_R	Scaled Time	$l_R^{0.5}$
ΔT_{oR}	Scaled Reference Temperature Rise	1
R_{oR}	Scaled Reference Radial Dimension	l_R
P_R	Scaled Thermal Power	$l_R^{0.5}$

Table 1. Scaling Ratios Summary Table

The total volume of the tank was scaled from the full plant by considering the scaled time response, assuming equal to 72 hours the full plant operating time of the RCCS in case of accident.

3. RELAP5-3D inputs description

The full scale and the experimental RCCS were modelled separately in RELAP5-3D. The hydrodynamic nodalization for the two plants is shown in Figure 3 [4].

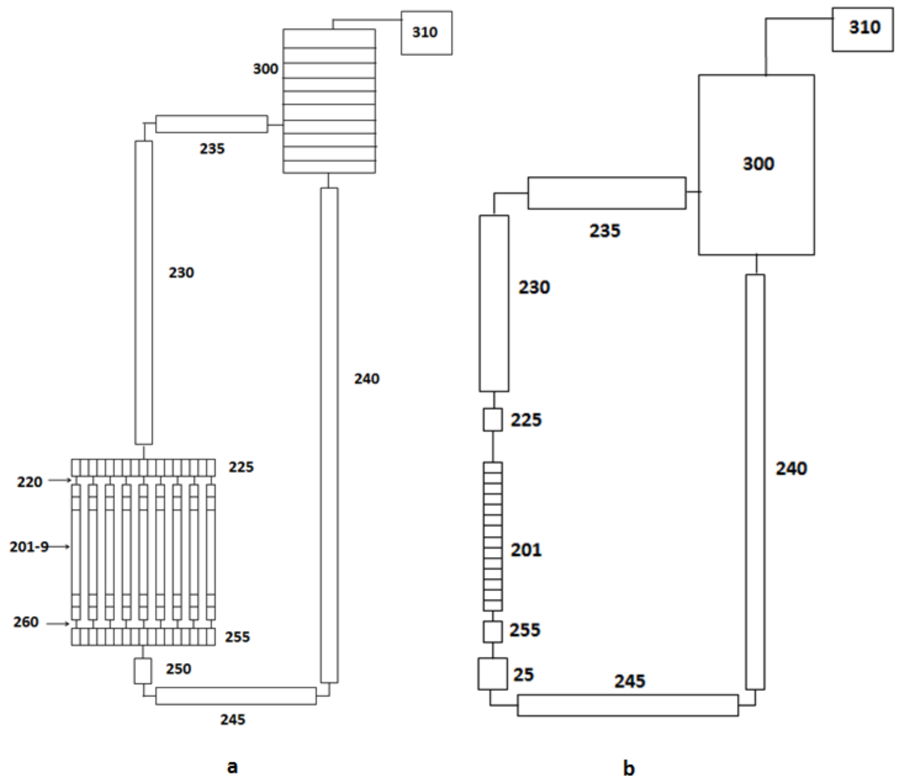


Figure 3. RELAP5-3D plants nodalization. (a. Experimental Facility; b. Full Plant)

The nodalization of the experimental facility is shown on Figure 3 a. The water tank on top of the facility was simulated with a pipe component (300) divided into 10 sub-volumes. Water flows down through the inlet pipeline which was modelled using three branch components (240, 245 and 250). The manifolds were modelled by horizontal pipe components (255 and 225) connected via multiple junctions to the nine risers. The risers were modelled individually using pipe components (201 to 209) which were specifically divided in 5 sub-volumes to match the portion inside the cavity (heated by the vessel) and the portion outside the cavity. This allowed a better analysis of the thermo-hydraulic phenomena occurring in the standing pipes over the length of each pipe and helped to overcome some limitations of the heat structures that will be described later. Two additional branch components were used to model the outlet pipeline connecting the upper manifold to the fourth sub-volume of the component 300. A similar nodalization was chosen for the full scale RCCS (Figure 3 b.). The main differences from the scaled facility nodalization are in the water tank, simulated by a single volume component (300) and in the risers which were all lumped into one single vertical pipe component (201). In both cases, a time dependent volume was connected to the top of the tank component to impose the ambient boundary conditions and to allow the vapour discharge during the transient. The nodalization of the scaled facility was further refined to account for heat exchange between the vessel and the pipes and the heat losses with the environment. In particular, back and front cavities of the facility were modelled by two single volumes (100 and 400 respectively) filled with air and connected to time dependent volumes to impose a constant ambient pressure as shown in Figure 4.

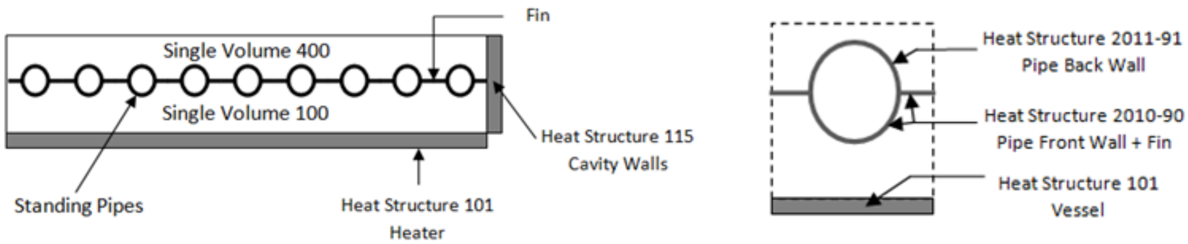


Figure 4. Experimental Facility Cavity Nodalization Top View and Heat Structures

Particular attention was dedicated to the heat transfer model. This is due to the relatively complex geometry and to some constraints of RELAP5-3D when modelling radiation, conduction and convection at the same time. The heat structures defined for the vessel, pipes and cavity walls are shown in Figure 4.

4. RELAP5-3D Steady-state simulations and scaling confirmation

Minor modifications to the input model described in the previous paragraph were adopted in order to reach the steady-state conditions. This includes an additional heat structure attached to the component 300 to remove the heat from the coolant, In order to avoid stratification that may increase the time required for the steady-state to be achieved, a single volume was used to model the tank. Different methods can be used to validate the scaling laws previously described. The approach used during the design process is summarized in the following diagram (Figure 5).

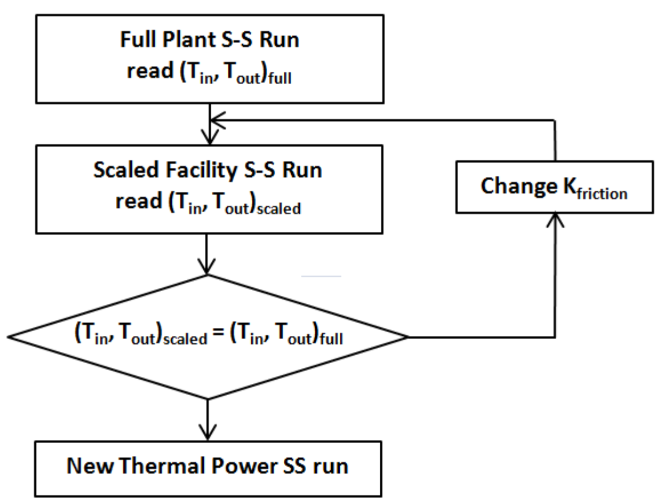


Figure 5. Scaling validation method flow chart

A Steady-State (S-S) calculation of the full power plant was performed and the temperature of the coolant at the inlet and outlet of the heated section of the risers was read. Then, the S-S calculation of the scaled facility model was conducted and the value of the temperatures at the corresponding locations were compared with the one found on the full power plant S-S calculation. During the first run, a discrepancy between the two sets of temperatures was found so the scaled facility calculation was repeated by adjusting the total mass flow rate until the set of temperatures at the defined locations matched the once for the full plant simulation. This was done by changing the form loss coefficient K of some of the junctions of the model (corresponding to elbows or change in flow area in the real facility). Table 2 shows the initial values of the form loss coefficient for selected junctions and the final values which produced the desired inlet/outlet riser's coolant temperature in the scaled facility model.

		Loss Coefficient (K)	
Junction #	Location	Initial	Final
24001	Tank Exit	0	200
24501	Tank-Bottom Manifold Elbow	0	100
25001	Bottom Manifold Inlet	0	200

Table 2. Junctions Loss Coefficient

A new set of calculations was carried out by applying different values of the total power in the reactor vessel of the full scale plant while the power of the scaled model was changed according to the defined scaling factor for the thermal power. Table 3 summarizes the results obtained with the design steady-state RCCS thermal power (0.7MW) and the expected value during DCC (1.5MW). An additional case at 2MW was also considered.

	0.7MW		1.5MW		2MW	
	Tin	Tout	Tin	Tout	Tin	Tout
Full	295	306	293.6 K	311.8 K	292.9 K	314.7 K
Scaled	294	305.2	293.6 K	311 K	293.4	314.4 K

Table 3. Steady-State Risers Inlet/Outlet Temperatures Summary

As expected, the change in the RCCS thermal power with the scaling laws defined did not produce appreciable change in the temperatures of the coolant at the risers between full plant and scaled facility, which was one of the main assumptions of the scaling procedures. The reference velocity can be related to the thermal power transferred to the water along the length of the risers and the water temperature rise as follows:

$$P_R = \dot{m} c_p \Delta T_{0R} = U_{0R} A_0 \rho_0 c_p \Delta T_{0R} \tag{4}$$

where $\dot{m}$ is the water mass flow rate, c_p, ρ_0 the reference specific heat and density of water, and A_0 the reference cross section of the risers. Assuming constant properties of the water, by Equation (4) it can be easily proven that the velocity scaling is automatically satisfied when the temperature rise is met.

5. RELAP5-3D Transient Analysis

A transient case was analyzed using RELAP5-3D. For this case a reference power of 1.5MW was applied (scaled with the appropriate laws described above). Since the system is open, the pressure of the facility is imposed by the atmospheric pressure. The initial static pressure evaluated at the bottom of the facility (component 245) was approximately 1.5e5 Pa. The analysis of the void fraction in the water tank during the transient has brought to the conclusion that the time response of the facility is strongly dependent on the void fraction of the mixture in the volume of the tank where the jet is located (Component 300, sub-volume 4).

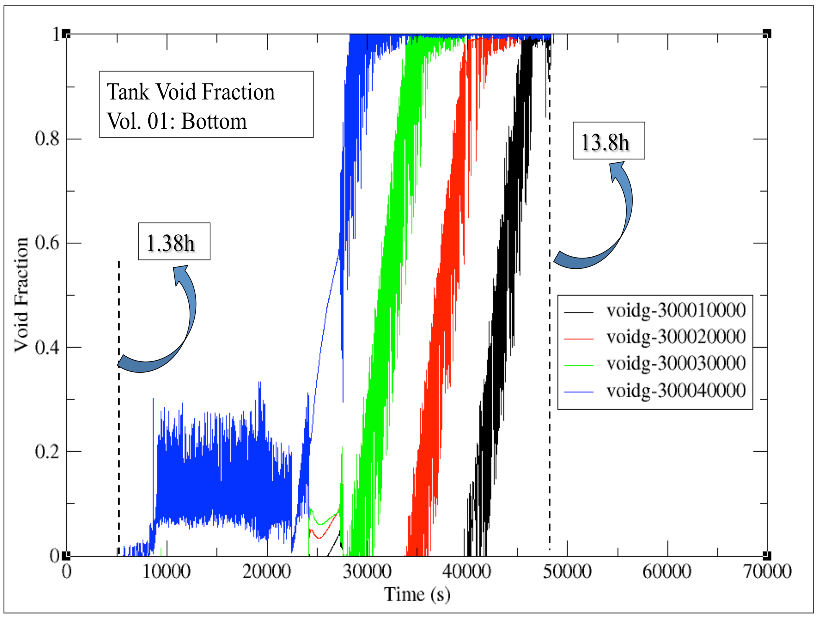


Figure 6. Water Mixture Void Fraction (Water Tank, Component 300)

In Figure 6 the void fraction of the subvolumes 1 to 4 of the component 300 is plotted. As one can see, void starts appearing in the sub-volume 4 at approximately 1.38 hours and then starts increasing with oscillations for the first 25000 seconds. At this point a faster slope was predicted while void starts appearing also in the volumes below the jet (sub-volumes3, 2 and 1). The time required to empty the water tank was found to be approximately 13.8 hours. The behavior of the total mass flow rate of the water in the system was found to be directly related to the void fraction transient previously described, as shown in Figure 7. In particular, oscillations in the mass flow rate were predicted to start when a small void fraction appears in the region of the tank near the inlet jet (volume 300-03). At approximately 25000 seconds from the beginning of the transient the liquid level in the tank goes below the jet position causing, as shown in Figure 7, a temporary flow inversion.

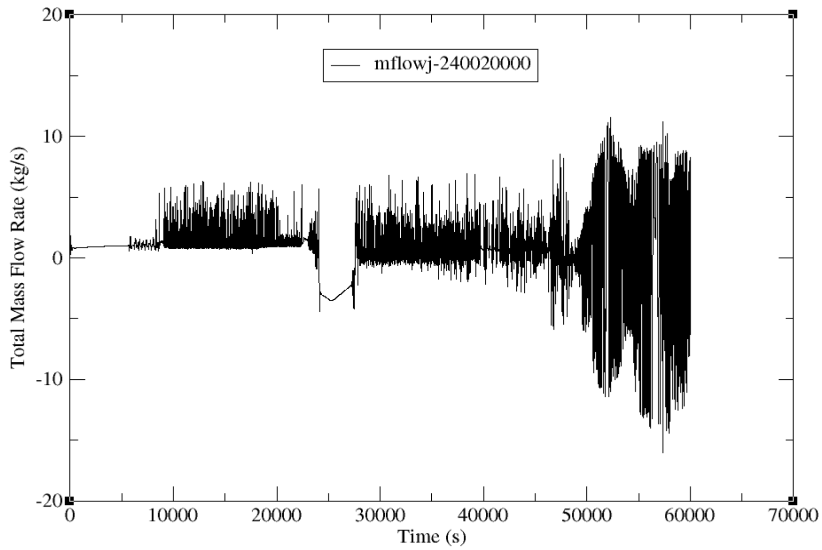


Figure 7. Total Mass Flow Rate (Tank Exit)

Also in this case oscillations were predicted. The period of these oscillations was confirmed to be in the order of 100 s, which is much longer than the time step used. Consequently, numerical instability was ruled out as the cause of the oscillations [5]. Same oscillations were found in on other parameters of interest such as coolant temperature and system pressure [6]. The analysis of the temperatures of the vessel wall showed a rapid increase of the temperature at approximately 14 hours, when most the water in the tank is completely evaporated. This time limit will be considered during the experimental activity in order to avoid any damage in the components due to overheating. The energy balance shown on Table 4 confirms the importance of the radiation heat transfer in the RCCS and highlight the needing to insulate the cavity of the experimental facility to limit the thermal losses.

	Power [W]	%
Total Power	12509.7	100.0
Convection to Pipes/Fin	1352.80	10.81
Radiation to Pipes/Fin	10593.98	84.69
Losses (rad + conv)	562.92	4.50

Table 4. Energy Balance over the Cavity

5.1 Top Manifold Sensitivity Analysis

A sensitivity analysis was conducted in order to verify whether recirculation of water between adjacent risers is possible. Recirculation was not predicted with the original nodalization described in the previous paragraphs. Different nodalizations of the top manifold (originally modeled with a horizontal pipe) were considered. The following results were obtained by modeling the top manifold with 3 horizontal pipes connected by cross junctions as shown in Figure 8.

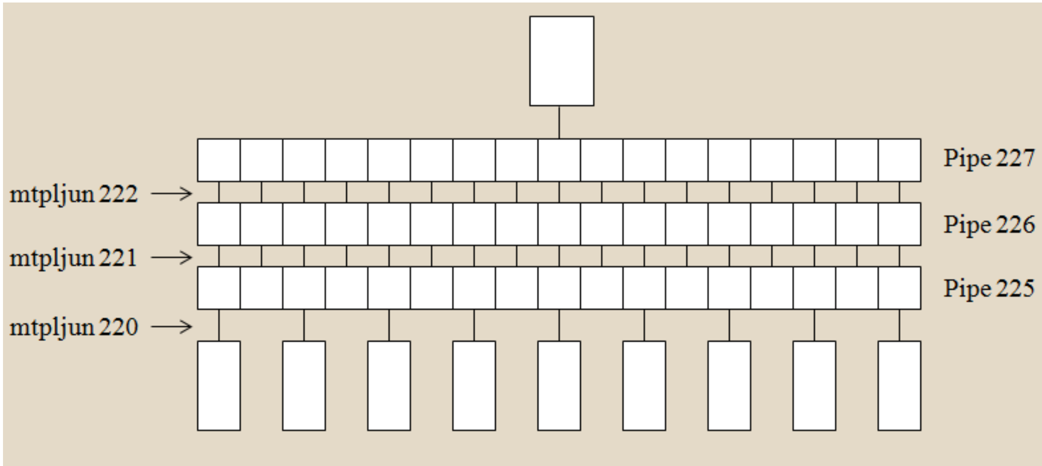


Figure 9. New Top Manifold Nodalization

This configuration allowed a better simulation of the liquid stratification in the top manifold. The mass flow rate through the risers 1 to 9 with this new configuration is shown Figure 10 together with the collapsed liquid level in the top manifold. Recirculation can be assumed to occur when the mass flow rate of two adjacent risers has opposite sign (positive=upwards, negative=downwards).

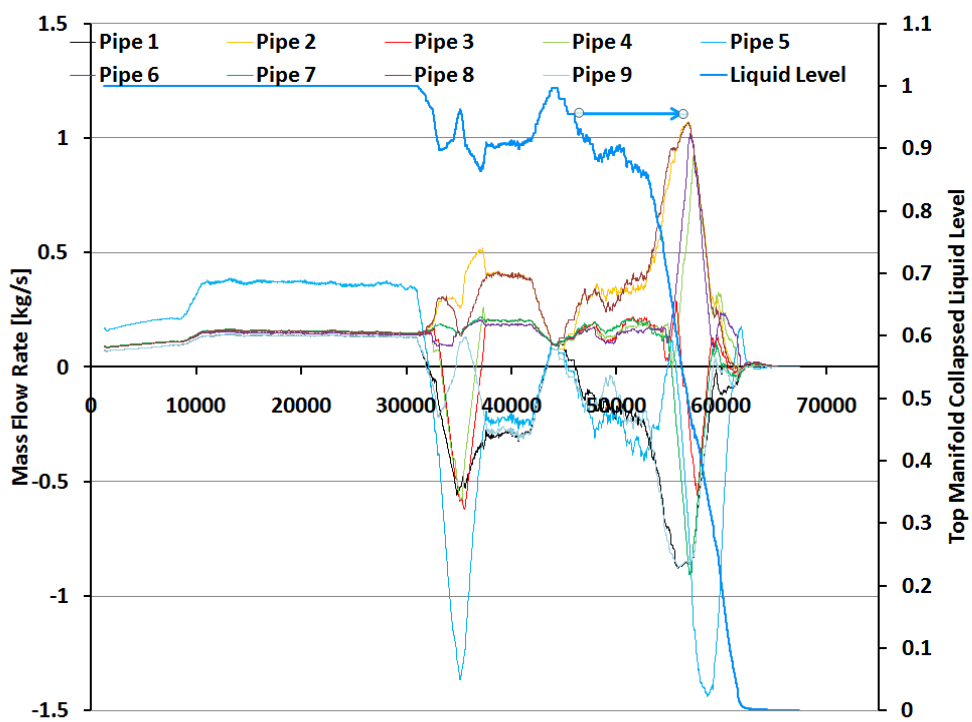


Figure 10. Risers Mass Flow Rates

Further analysis of the mass flow rates through the risers during the first phase of the transient (t<30000s), show that the highest mass flow rate is estimated to occur at the center risers (pipe 5) due to its relative position from the manifold's inlet while the lowest is estimated to occur in the risers placed at the edge of the cavity (pipes 1 and 9), far from the manifold's inlet.

6. CFD Simulation Description and Results

Based on the RELAP5 simulation, the overall facility behavior can be divided into four phases such as sub-cooled liquid, saturated (small and large void fraction) and saturated vapor. In the present paper the analysis of the very first phase of the transient is shown. CFD simulations were carried out in order to study local phenomena inside the risers and the manifolds that are difficult to simulate with system codes. This analysis was based on the steady state of one particular time step taken from the sub-cooled phase. The portion of the system modeled includes the top and bottom manifolds and the nine risers. The boundary conditions required for the simulation, such as coolant inlet and outlet temperatures and void fraction, inlet mass flow rate, outlet pressure and risers heat flux were taken from the RELAP5-3D calculations at t=1200s . Particular attention was given to the heat flux to account for the non-uniform flux between the nine risers, the heated and unheated sections of the walls and the different conditions between front and back walls of the risers.. Figure 11 shows the geometry simulated and the boundary conditions imposed.

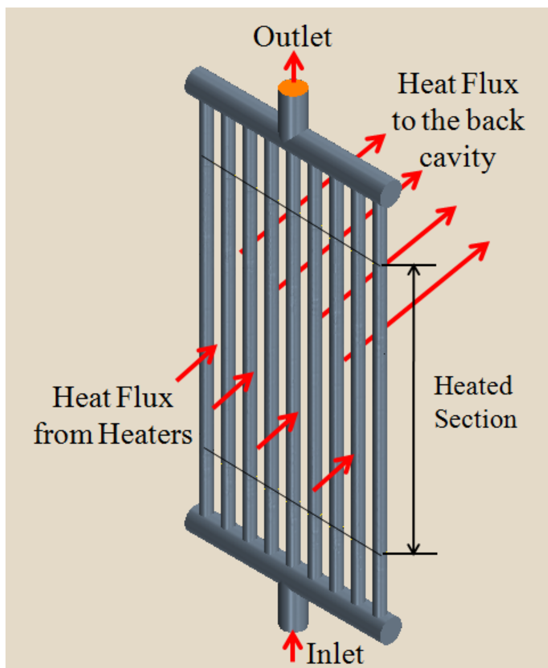
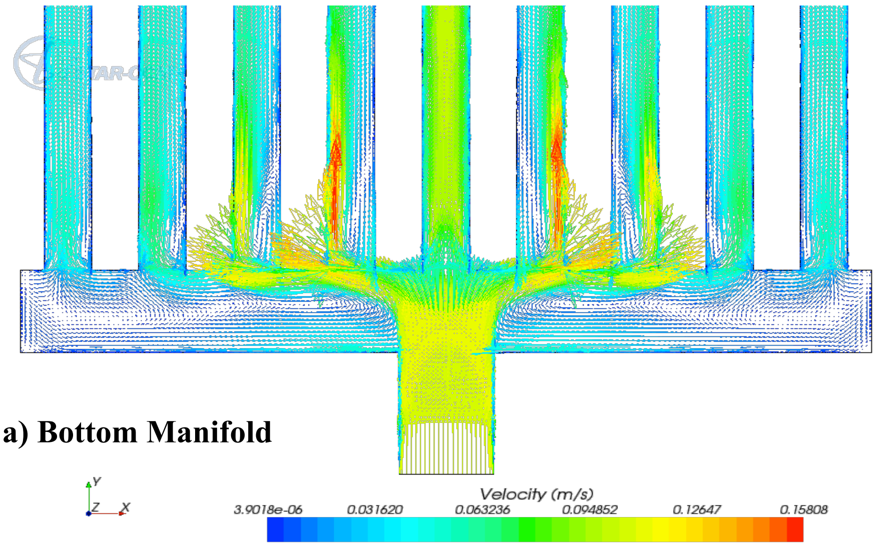
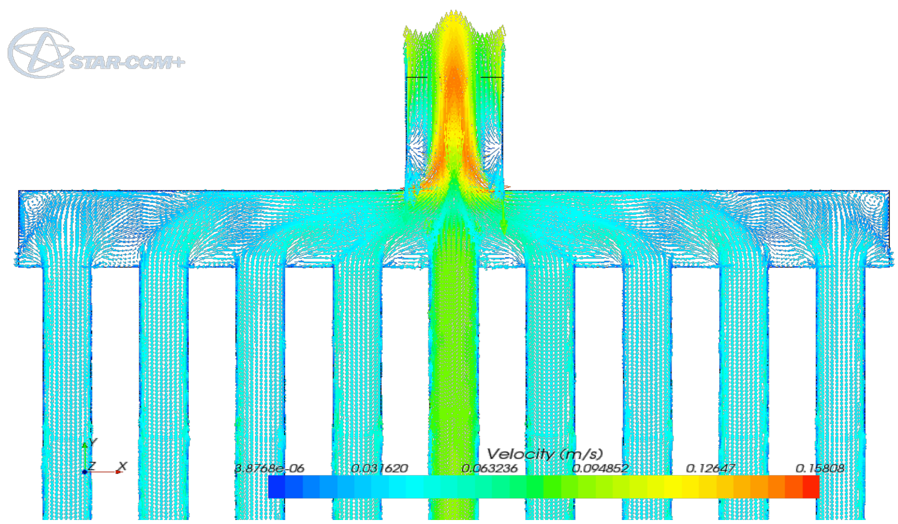


Figure 11. CFD Geometry and Boundary Conditions

The number of meshes used for the simulation was 1.15 million with an hexagonal mesh size of 0.5 cm. The Realizable K-ε turbulent model was selected through the study with the Reynolds-Average Navier- Stokes equation. One of the most important and remarkable results can be seen by observing the velocity profile of the coolant in the bottom and top manifolds, as shown in Figure 12.



a) Bottom Manifold



b) Top Manifold

Figure 12. Velocity Profile

The largest velocity is predicted to be at the center riser, due to the relative position with the inlet/outel, and gradually decrease from the center to the edge. A high vorticity region is predicted at the entrance of the risers especially for the ones near the manifold inlet (Figure 11, a.). This phenomenon may cause a higher local pressure drop and reduce the mass flow rate through these risers. Far from the manifold inlet, this vorticity is negligible. Recirculation was observed in the top manifold in the regions between the inlets, and on the sides of the main outlet. The mass flow rate distribution through the nine risers predicted by the CFD code is in good agreement with the prediction of the RELAP5-3D system code as shown in Figure 13, where the small differences were identified to be related to the differences in the models between the two codes. In the same figure, a comparison between two different mesh sizes (10mm and 0.5mm) is also shown.

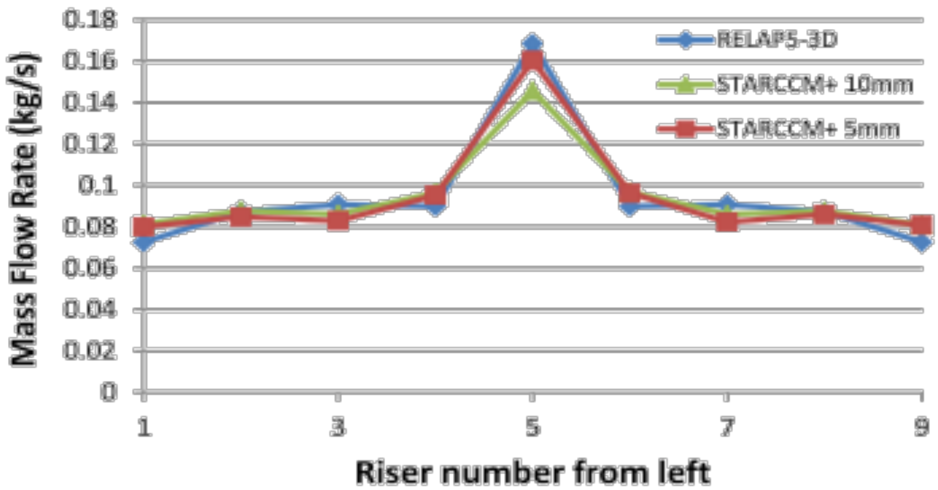


Figure 13. Mass Flow Rates Prediction.

7. Conclusions

The procedure and laws applied to scale down the Texas A&M RCCS experimental facility were confirmed by comparing the steady-state results of the RELAP5-3D simulations. These simulations revealed the importance of changing the total friction loss of the facility at the beginning of the experimental facility in order to match the inlet/outlet riser's coolant temperature which must be monitored during the experiments. The overall thermohydraulic behavior facility was predicted using RELAP5-3D. The predictions showed some important features of the coolant flow such as oscillations that must be taken into account during the experimental observations and measurements. The total time required with the imposed power to empty the water tank was found to be approximately 13.8 hours which will be used as a time limit to avoid damage of the component due to high temperatures. RELAP5-3D was also able to predict recirculation between adjacent risers when the model of the top manifold was opportunely modified. Void fraction in the water tank must be directly observed during the experimental activity since it was found to drive the thermodynamic behavior of the mixture in the facility. The predictions obtained with the CFD code focus the attention to the bottom portion of the facility, during the first phase of the transient where high vortex may be observed. The predictions of RELAP5-3D are in agreement with the overall expected behavior of the system and confirmed the potentiality of the RCCS to passively remove the decay heat from the cavity during the design period.

8. Acknowledgment

The authors would like to mention Luigi Capone for his contribution to the facility scaling and for his guidance on the CDF simulations. NEUP funding is greatly appreciated. We gratefully acknowledge the use of EOS computing facility at Texas A&M University for their support and computing hours for this study.

9. References

[1] RELAP5-3D Code Manuals, INEEL-EXT-98-00834, Rev. 2.4.

[2] H.J. van Antwerpen, G.P. Greyvenstein, "Evaluation of a detailed radiation heat transfer model in a high temperature reactor systems simulation model", *Nuclear Engineering and Design* 238 (2008) 2985–2994.

[3] D. D. Lisowski, S. M. Albiston, R. M. Scherrer, T. C. Haskin, M. H. Anderson, A. Tokuhiro, M. L. Corradini, "Experimental Studies of NGNP Reactor Cavity Cooling System with Water", Proceedings of ICAPP 2011 Nice, France, May 2-5, 2011 Paper 11116.

[4] R. Vaghetto, Y.A.Hassan, "Analysis of the Thermo-Hydraulic Behaviour of the New Texas A&M RCCS Experimental Facility Using RELAP5-3D", Proceedings of ICONE19, 19th International Conference on Nuclear Engineering May 16-19, 2011, Chiba, Japan.

[5] P.D.Bayless et al., "Severe Accident Natural Circulation Studies at INEL", NUREG/CR-6285 INEL-94/0016.

[6] Guo Yun, Zeng Heyi, "Instability in Parallel Channel System", Proceedings of 3rd US_European Fluids Engineering Summer Meeting and 8th Int. Conference on Nanochannels, microchannels and minichannels, FEDSM2010-ICNMM2010. August 1-4, 2010 Montreal, Canada.