Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 22, 2009

A Study on Heat-up Phenomena of the RHR Pump Room in KORI Unit 2 for the Loss of HVAC Accidents

Churl Yoon *, Jinhee Park, Ho Gon Lim, Sang Hoon Han
Korea Atomic Energy Research Institute 150 Dukjin-dong, Yuseong, Daejeon, KOREA
**Corresponding author: cyoon@kaeri.re.kr*

1. Introduction

In PSA(Probabilistic Safety Analysis) Models, the HVAC(Heating, Ventilation, and Air Condition) system is essential for the various vital mitigation safety systems operating during a mission time. So far, the unavailability of a safety system when the HVAC system fails has been applied conservatively or optimistically based on operating experience and expert judgment, so the total core damage frequency could be unrealistic. When the HVAC system of a nuclear power plant fails, it is one of the main issues in a PSA FT (Fault Tree) model to estimate the transient temperature variations of some component rooms. The purpose of this study is to establish a heat-up prediction model by using direct measuring and CFD(Computational Fluid Dynamics) analyses.

2. Methods and Results

Direct measurement and a CFD analysis were performed for the heat-up of a RHR(Residual Heat Removal) pump room in KORI Unit 2 under the condition of HVAC loss.

2.1 Direct Measurement

The error in a CFD analysis consists of modeling errors, discretization errors, and the other errors including round-off error and programming bugs. Evaluation of these errors should be performed separately to prevent that different errors could compensate each other. Especially to evaluate the modeling error that is generally the largest one, direct measuring room temperatures was performed.

The RHR pump in KORI Unit 2 had been run for about four hours without any of HVAC system working. During the test period, temperatures were measured by K-type thermal-couples (Omega Co.) at various locations. Table I summarizes the measuring locations, in which the origin was assumed at the north bottom corner of the pump room. The thermal-couples are calibrated against a reference thermo-meter with an error of ± 0.04^{o}C. Figure 1 shows the transient temperature measurement. At a relative time of 0, the RHR pump motor was started without any HVAC working. At relative time of ~ 3hr 55min, the HVAC system including a cubical cooler was started. During the test, the corridor temperature outside the pump room was measured to be constant 26.5^{o}C.

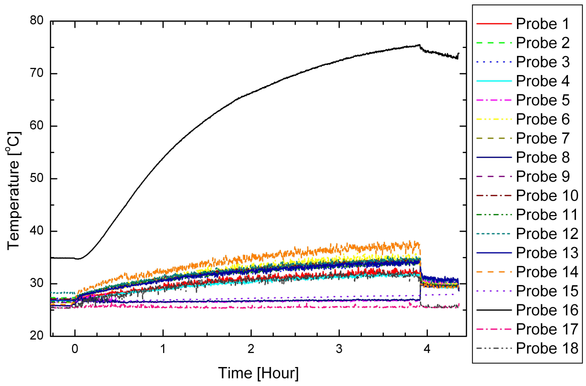


Fig. 1. Measured Temperatures at Various Locations in a RHR Pump Room of KORI Unit 2 during 4-Hours Pump Run without HVAC.

Table I: Measured Locations

Probe No.	Location (x, y, z) in cm	Probe No.	Location (x, y, z) in cm
1	(40, 635, 383)	10	(340, 110, 500)
2	(80, 580, 505)	11	(440, 290, 340)
3	(385, 580, 375)	12	Above the Motor
4	(385, 550, 507)	13	(192, 350, 430)
5	(130, 90, 100)	14	(440, 290, 220)
6	(130, 90, 300)	15	Inner Wall Surface
7	(130, 90, 500)	16	Motor Surface
8	(340, 110, 100)	17	Exit Upper
9	(340, 110, 300)	18	Exit Lower

2.2 Numerical Simulation

Computational Fluid Dynamics (CFD) is applied to estimate the room air temperature, instead of conventional lumped methods. A commercial CFD code, ANSYS CFX release 10, is used for the simulation. Figure 2 shows the mesh geometry and temperature probing locations in the CFD analysis. The computational domain includes concrete room walls, a motor as a heat source (pink block in Fig. 2), and a cubical cooler as an obstacle (green block in Fig. 2). The obstacles inside the pump room such as the pipes, electric devices and lines, and compressor are omitted. These obstacles could disturb the air circulation, which enhances advection heat transfer. On the other hand, a structure is mostly made by metal that could work as a heat sink. Therefore, the omission is considered conservative. The meshes are structured grids with 94,363 cells or 52,394 cells.

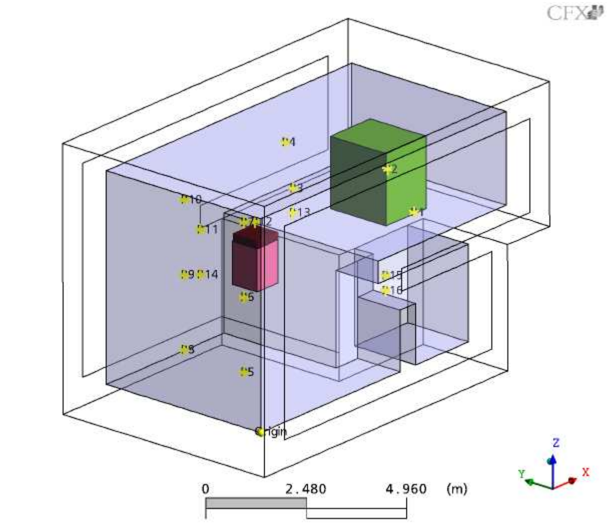
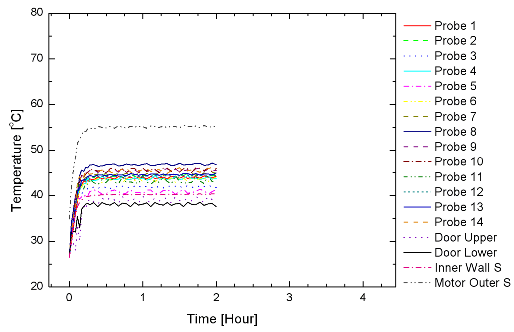


Fig. 2. Mesh Geometry and Probing Locations used for CFD Analysis.

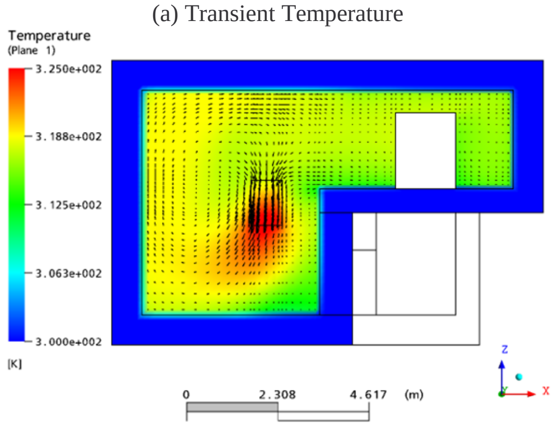
The heat source was assumed to be the motor only, and approximated as a porous block. The formal capacity of the motor is 200 HP, and the efficiency and the power factor are respectively 92.6% and 82.6%. The calculated pump work is about 150 HP, so that the maximum heat generation rate becomes 50 HP (37.285 kW) if all the other energy except for the pump work were converted to thermal energy. Figure 3 shows simulation results for a 30 HP heat load. The main difference from the previous CFD model [2] is that the existence of a cooling fan above the motor was observed in the direct measurement and that a momentum source term was added on the top of the motor in the CFD model to produce cooling flows through the motor.

The three major affective factors in this CFD are heat generation rate, cooling fan momentum, and porosity of the assumed porous medium. Sensitivity studies have been performed for these major factors. As a result, porosity affects the results less and heat generation rate is found to be the most effective element.

By comparing the measured and the predicted temperatures, it is observed that the CFD model over-predicts the room temperature at the early stage of the transient. In a real motor, heat is mostly generated in metallic coil and a shaft and the generated heat increases the metal temperature first. And then, convective heat transfer would not occur until the temperature difference between the surrounding air and the motor structure becomes a certain amount. But in this CFD model, metallic structure of the motor was omitted and only the air flows through the motor was modeled by porous media approximation, because modeling the complex metallic part of a motor is very difficult in this CFD analysis with the relatively large length scale. Therefore, we can concludes that ignorance of the latent heat of the motor structure and the time duration of metallic heat-up results in the over-estimation of room air temperature at the early stage of the accident transient.



(a) Transient Temperature



(b) Temperature Distribution at 2 Hour

Fig. 3. Simulation Results with a Heat Generation of 30HP.

3. Conclusions

To establish a CFD analysis model for a heat-up prediction of a reactor component room for the loss of HVAC accidents, direct measurements and CFD predictions have been performed for a RHR pump room in KORI Unit 2. By comparing the measured and the predicted temperatures, it is found that the largest modeling error exists at the early stage of the transient. The results of this study could be applied to the PSA Model for a Risk Informed Regulation for KORI Unit 2.

REFERENCES

[1] NUMARC 87-00, Guidelines and Technical Bases for NUMARC Initiatives Addressing Station Blackout at LWRs

[2] C. Yoon, M. Hwang, J.E. Yang, and J. Park, “Heat-up Calculation of the Auxiliary Feed Water Pump Room at the Ulchin Units 3 & 4 for Loss of HVAC Accidents”, 17th Korea ANSYS User’s Conference, Cheongju, Korea, Nov. 8~9, 2007.

[3] CFX-5: Solver Theory, ANSYS Canada Ltd., Canada, 1996.

[4] Adrian Bejan, Convection Heat Transfer, 2nd Ed., Chap. 6, john Wiley and Sons, Inc., 1995.

[5] P.M. Daling, J.E. Marler, T.V. Vo, H. Phan, J.R. Friley, Value Impact Analysis of Generic Issue 143, “Availability of Heating, Ventilation, Air Conditioning (HVAC) and Chilled Water System”, App. F, NUREG/CR-6084, 1993.