Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 26-27, 2017

Methodology for Thermal Analysis of Spent Fuel Dry Storage Cask Design using MARS Code

Kyung Mo Kim, Yeong Shin Jeong, In Cheol Bang *
Department of Nuclear Engineering,
Ulsan National Institute of Science and Technology (UNIST),
50 UNIST-gil, Ulju-gun, Ulsan 44919
**Corresponding author: icbang@unist.ac.kr*

1. Introduction

Disposal of the spent fuel, which generates decay heat and contains radioactive material, becomes an essential ingredient due to saturation of spent fuel storage pool and expiration of established nuclear power plants. Although technologies related to reprocessing and further use of the spent fuel are under development for the disposal of spent fuels, the spent fuel must be managed in appropriate manners until the further use.

Therefore, interim spent fuel storage cask has been developed in the form of dry storage. The casks have been designed to meet the safety acceptance criteria provided by US NRC in terms of structural, thermal, confinement, shielding, criticality, and material against various off-normal conditions and postulated accident scenarios. The passive decay heat removal capacities of the designed spent fuel dry storage casks have been quantified by experiments with scale-down mock-ups and numerical simulations with computational fluid dynamic (CFD) codes. Many research [1-5] also analyzed the thermal behavior inside the cask regarding various conditions by commercial CFD codes and experiments.

However, the thermal analysis during the transient condition through CFD codes require much time, and the obtained results are sensitively dependent on many parameters such as mesh cell number, boundary conditions, and applied models. Besides, realization of the transient conditions in the experiments with scale-down mock-up or full-scale product is difficult.

Thus, one-dimensional transient analysis code regarding thermal-hydraulics behavior in a large size system, MARS (multi-dimensional analysis for reactor safety) code, was selected as simulation tool for spent fuel dry storage casks. The establishment of thermal analysis methodology will be meaningful in a viewpoint of transient calculation of various off-normal and accident scenarios. Design improvement of spent fuel dry storage cask also can be achieved through the time-saving on thermal and safety analysis having similar level of prediction accuracy compared to the general tools.

In this study, experimental work and MARS simulation were peformed with comparison of their results to find the best-estimate analysis method regarding the thermal behavior of the spent fuel dry storage cask.

2. Experimental

Thermal behavior of the spent fuel dry storage cask in normal condition was observed through experiment with scaled mock-up to produce data, which will be compared with simulation results by MARS code to establish a best-estimate analysis method.

2.1 Experimental Setup and Condition

A mock-up was prepared to be scaled-down to 1/10 of dimension of metal cask, which is developing by KORAD as shown in Fig. 1. Total 21 baskets simulating the spent fuel assemblies were equipped inside the canister. The decay heat of the spent fuel assemblies was simulated by cartridge heaters embedded inside the baskets. Cask wall having larger diameter than that of canister surround the canister, and air was filled inside the canister and gap between canister and cask walls. The canister lid and impact damper at bottom and top side enclosed the cask.

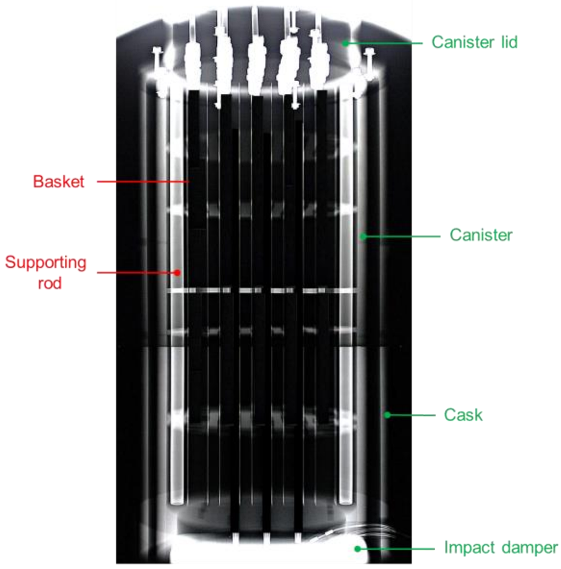

Fig. 1. Composition of 1/10 scaled mock-up.

After preparation of the mock-up, 0.1 kW (an equal linear power density of spent fuel assemblies in KORAD metal cask) was loaded on the cartridge heaters to realize the normal operating condition of commercial cask. During the experiment, variations of the air temperatures inside the canister and gap between cask and canister walls were measured by K-type thermocouples.

Table I: Comparison of 1/10 scale-down mock-up dimensions and metal cask of KORAD

Components	Dimensions [mm]	
	Metal cask (KORAD)	Mock-up (UNIST)
Cask body	Φ2126 x 5285	Φ210 x 530
Basket	241 x 241 x 4550	20 x 20 x 450
Canister	Φ1686 x 4880	Φ170 x 480
Q/basket (kW)	1.0	0.1
Q'/basket (kW/m)	0.22	0.22

2.2 Results and Discussion

The variations of air temperatures during the experiment were plotted in Fig. 2. Air temperatures inside the canister were around 110 °C. As the elevation increases, the air temperature increased gradually owing to natural circulation inside the canister, which is caused by density difference. The air inside the gap between canister and cask walls at steady state was 40 °C, which is slightly lower than that inside the canister. Wall temperatures of cask and canister lid changed similarly to each other because the radiation and convection through radial direction is a cooling path of the cask, while the canister lid showed higher temperature compared to cask wall at steady state because of natural circulation effect. Phenomenological insights on the thermal behavior inside the cask, such as natural circulation, radiation, and convection heat transfer, were obtained through the temperature measurements during the experiment.

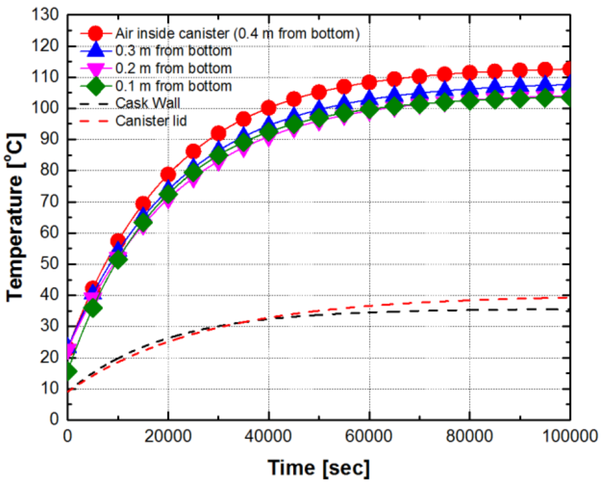

Fig. 2. Measured air temperature variations during the experiment.

3. Numerical Analysis using MARS Code

The experimental work, which was conducted with scale-down mock-up, was simulated by MARS code to analyze the prediction capability of the code and to find best-estimate analysis method.

3.1 Analysis Model

The nodalization of experimental mock-up is drawn in Fig. 3. The 21 baskets were modeled as a heat structure having bundle option (pitch to diameter ratio of 1.25). The internal heat source model was used to simulate the heat load on cartridge heaters. The air flow channel inside the canister was modeled with branch components for the analysis of cross-flow. Convection boundary condition was given at canister and cask walls to simulate the heat transfer through natural convection of the air. The atmosphere, which is ultimate heat sink of the cask, had constant temperature condition. Bottom side of impact damper was adiabatic condition, whereas the heat transfer at the upper part of canister lid was modeled to be occurred in manner of natural convection. The transient calculation was continued until the achievement of steady state.

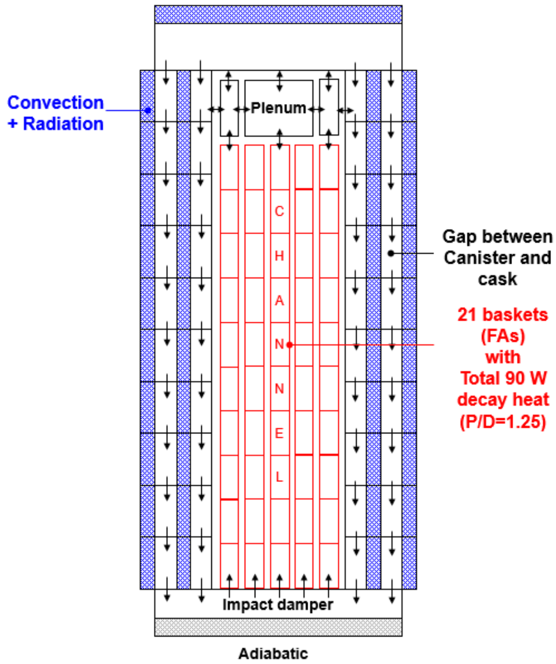

Fig. 3. MARS nodalization of scaled mock-up.

3.2 Establishment of Best-Estimate Analysis Method

In simulation with default model, any options related to the heat transfer were not used to evaluate the prediction capability of the code on thermal analysis of gas system. Radiation enclosure option was also neglected. Only natural convective heat transfer was calculated by Churchill and Chu [6] correlation, which has form of Eq. (1).

$$h_{convection} = \frac{k\left\{0.825 + \frac{0.387\, Ra_L^{1/6}}{\left[1+(0.492/\mathrm{Pr})^{9/16}\right]^{8/27}}\right\}^2}{H} \quad (1)$$

However, Churchill-Chu correlation was developed in geometry of vertical flat plate, consequently, the geometric effect of cask and canister (cylinder) on natural convective heat transfer could not be analyzed accurately. Therefore, several models, which were suggested to predict the natural convective heat transfer at the cylinder surface, were surveyed as presented in Table II. Among the surveyed models, Yang`s correlation, which modified the Churchill-Chu correlation by considering the effect of ratio between height and diameter of cylinder, was selected as an optimal model for the prediction of natural convective heat transfer because other correlations were validated in the condition of constant temperature, and have limited applicable range of Grashof number and Raynolds number, while Yang`s model was validated with experimental data of constant heat flux condition and complete range of Gr and Ra. In addition, radiation heat transfer was considered in the best-estimate analysis method because the radiation become important heat transfer mechanism when the convective heat transfer coefficient is low and surface temperature is significantly high. For the consideration of radiation heat transfer, the radiation heat transfer coefficients according to surface temperature was calculated by Eqs. (2) and (3). Consequently, the heat transfer coefficients at the walls were calculated by summation of the heat transfer coefficients from Eq. (2) and Yang`s correlation.

$$Q_{rad} = \varepsilon\sigma A\left(T_{sur}^4 - T_f^4\right) = h_{rad} A(T_{sur} - T_f) \quad (2)$$

$$h_{rad} = \frac{\varepsilon\sigma\left(T_{sur}^4 - T_f^4\right)}{\left(T_{sur} - T_f\right)} \quad (3)$$

$$Q_{total} = \left(h_{rad} + h_{convection}\right)A(T_{sur} - T_f) \quad (4)$$

The temperature variations analyzed by default model and best-estimate method were compared with each other as shown in Fig. 4. The experimentally measured air temperature distribution at steady state was compared with analysis results by default model, and best-estimate method as presented in Table III. The simulation considering the effects of geometry on natural convective heat transfer and radiation heat transfer showed better prediction capability compared to simulation results with default model. The difference between temperature distributions of experiment and analysis by best-estimate method would be caused by uncertainties of cross-flow calculation in basket bundle and used heat transfer model regarding natural convection, and neglection of view factor in terms of radiation heat transfer.

Table II: Summary on models predicting natural convective heat transfer at outer surface of cylinder

Correlations [7-10]
$\frac{Nu_H}{Nu_{H,FP}} = 1 + 0.3\left[32^{0.5}\, Gr_H^{-0.25}\, \frac{H}{D}\right]^{0.909}$ (Cebeci, 1974)
$Nu_H = \left[A + B\left(\frac{H}{D}\right) + C\left(\frac{H}{D}\right)^2 + E\left(\frac{H}{D}\right)^3\right] \times Ra_H^{0.25 - F\frac{H}{D} + G\left(\frac{H}{D}\right)^2}$ * $A = 0.519$, $B = 0.03454$, $C = 0.0008772$, $E = 8.855 \times 10^{-6}$, $F = 0.00253$, $G = 1.152 \times 10^{-5}$ (Popiel et al. 2007)
$Nu = \frac{4}{3} Ra_H^{0.25}\left[\frac{7\,\mathrm{Pr}}{100 + 105\,\mathrm{Pr}}\right]^{0.25} + \frac{4}{35}\frac{272 + 315\,\mathrm{Pr}}{64 + 63\,\mathrm{Pr}}\frac{H}{D}$ (Le Fevre and Ede, 1956)
$Nu_H^{0.5} = 0.6\left(\frac{H}{D}\right)^{0.5} + 0.387\left(\frac{Ra_H}{\left[1 + (0.492/\mathrm{Pr})^{9/16}\right]^{16/9}}\right)^{1/6}$ (Yang, 1985)

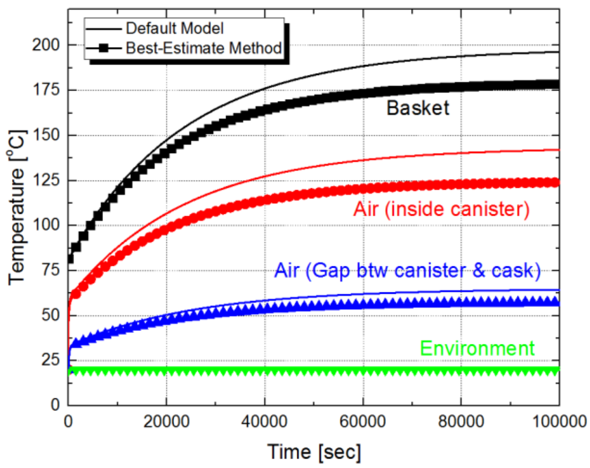

Fig. 4. Comparison of temperature variations calculated by default model simulation and best-estimate method.

Table III: Comparison of air temperatures according to positions at steady states

Position	Temperature [°C]		
	Exp.	Default analysis	Best-estimate analysis
Inside canister	112	142	125
Gap btw canister and cask	40	65	50
Environment	20	20	20

Based on the established best-estimate analysis method, various parameters, such as air velocity profile, pressure, and criticality, could be analyzed in the transient conditions as shown in Figs. 5 and 6. As the elevation increases, the air velocity increases because due to the temperature profile inside the canister. Internal pressure also increases as the canister temperature increases. The mentioned parameters are during the off-normal or accident conditions are difficult to predict using commercial CFD or neutronics codes, despite those are crucial factors in evaluating the integrity of the cask. Hence, suggest analysis method in this paper will be useful in the design and safety analysis of spent fuel dry storage cask, reducing the time and expenses, compared to previous methodologies.

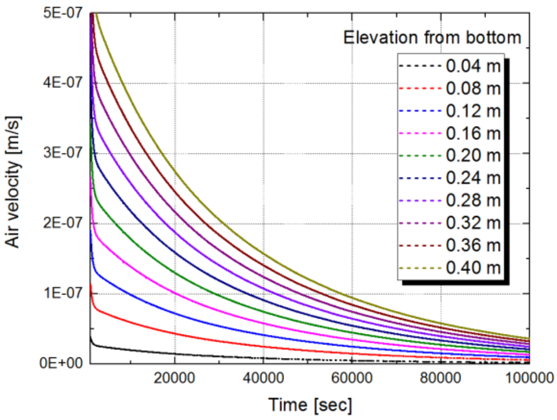

Fig. 5. Calculated natural convection flow rate of air inside canister according to elevations.

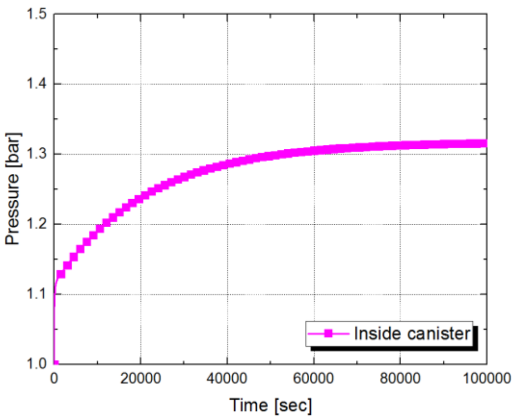

Fig. 6. Calculated pressure variation inside the canister.

4. Conclusions

For the analysis of thermal behavior inside the spent fuel dry storage cask, MARS code, which is one-dimensional system analysis code was used in this study. Thermal analysis results, obtained from MARS code simulation, were compared with experimental work using scale-down mock-up, for the evaluation of prediction capability of the code according to analysis methodologies. The analysis methodology considering the effects of cylinder geometry on natural convective heat transfer, and radiation heat transfer showed better prediction capability than the simulation results based on default models in terms of air temperature distribution at steady state. It is expected that the best-estimate methodology, suggested by this study, will be more efficient safety analysis method, compared to the previous analysis tools, because it reduces calculation time required to the simulation of transient conditions such as off-normal and accident with good prediction accuracy.

ACKNOWLEDGEMENTS

This work was supported by the Nuclear Energy Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Science and ICT. (No. NRF-22A20153413555,2016M2A8A6900481, 2013M2B2B1075734)

REFERENCES

[1] S. W. Churchill, and H. H. S. Chu, Correlating Equation for Laminar and Turbulent Free Convection from Vertical Flat Plate, International Journal of Heat and Mass Transfer, Vol.18, p. 1323, 1975.
[2] K. S. Bang, S. H. Yu, S. H. Lee, J. C. Lee, and K. S. Seo, Experimental Investigation of heat removal performance of a concrete storage cask, Annals of Nuclear Energy, Vol.85, p.679, 2015.
[3] H. Kim, O. J. Kwon, G. U. Kang, D. G. Lee, Comparisons of Prediction Methods for Peak Cladding Temperature and Effective Thermal Conductivity in Spent Fuel Assemblies of Transportation/storage Casks, Annals of Nuclear Energy, Vol.71, p.427, 2014.
[4] H. Kim, D. G. Lee, G. U. Kang, C. H. Cho, and O. J. Kwon, Analysis of Heat Transfer on Spent Fuel Dry Cask During Short-term Operations, Journal of Computational Fluids Engineering, Vol.21, p. 54, 2016.
[5] R. L. Frano, G. Pugliese, and G. Forasassi, Thermal Analysis of a Spent Fuel Cask in Different Transport Conditions, Energy, Vol.36, p. 2285, 2011
[6] S. W. Churchill, and H. H. S. Chu, Correlating Equation for Laminar and Turbulent Free Convection from Vertical Flat Plate, International Journal of Heat and Mass Transfer, Vol.18, p. 1323, 1975.
[7] T. Cebeci, Laminar-Free-Convective-Heat Transfer from Outer Surface of Vertical Slender Cicular Cylinder, Proceedings of the 5thInternational Heat Transfer Conference, 1974, Tokyo.
[8] C. O. Popiel, J. Wojtkowiak, K. Bober, Laminar Free Convective Heat Transfer form Isothermal Vertical Slender Cylinders, Experimental Thermal and Fluid Science, Vol. 32, p. 607, 2007.
[9] E. J. Le Fevre, A. J. Ede, Laminar Free Convection from the Outer Surface of a Vertical Circular Cylinder, Proceedings of the 9th International Congress of Applied Mechanics,1956, Brussels.
[10] S. M. Yang, General Correlating Equations for Free Convection Heat Transfer from a Vertical Cylinder, Proceedings of International Symposium on Heat Transfer, 1985, Peking.