Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 6-8, 2026

Development and Validation of Thermal Analysis Model for Fuel Assembly Canister Test Simulator(FACTS)

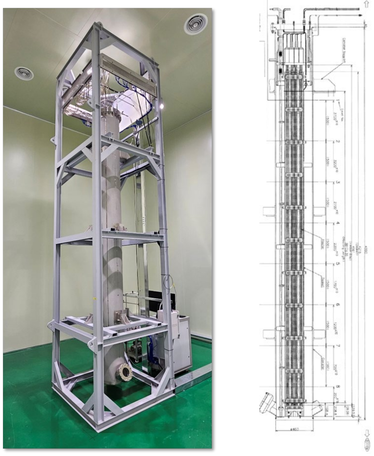
Do Yun Kim[a*], Ju-chan Lee[a], Seunghwan Yu[a]
[a] Transportation and Storage R&D Division, Korea Atomic Energy Research Institute,
Daejeon 34057, Republic of Korea
dykim0129@kaeri.re.kr

***Keywords : Dry storage, Fuel Assembly Canister Test Simulator(FACTS), CFD, COBRA-SFS, Thermal test.**

1. Introduction

The long-term management of spent nuclear fuel (SNF) necessitates the implementation of reliable dry storage systems. A primary safety requirement for these systems is maintaining the peak cladding temperature (PCT) below the regulatory limit of 400°C to prevent the degradation of cladding integrity. Traditionally, thermal evaluations have relied on conservative modeling approaches to ensure sufficient safety margins. However, there is an increasing demand for best-estimate evaluations to optimize storage capacity and reduce operational costs. Such high-fidelity evaluations require validation of numerical tools against high-quality experimental data.

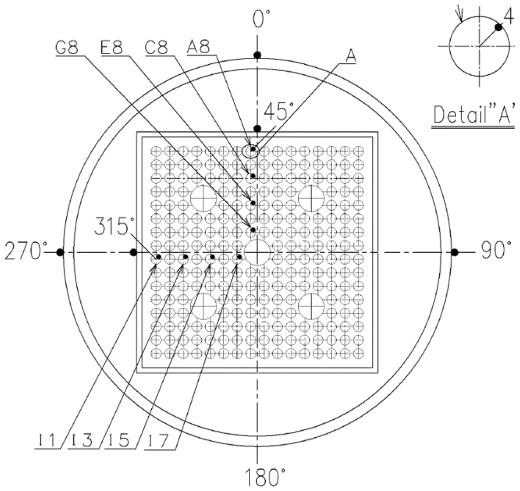
In this context, the Fuel Assembly Canister Test Simulator (FACTS) facility was developed as a domestic experimental platform to simulate the thermal-hydraulic behavior of SNF during dry storage and vacuum drying processes. Utilizing a commercial-scale PLUS-7 fuel assembly skeleton, the FACTS facility provides a unique environment for producing precise temperature data. This paper presents the results of thermal characteristic evaluations and the subsequent validation of Computational Fluid Dynamics (CFD) and COBRA-SFS models based on the experimental findings from the FACTS facility.



[Fig. 1. Schematic layout and instrumentation of the FACTS facility]

2. Experimental Setup and Numerical Methodology

The FACTS facility is specifically engineered to replicate the environmental conditions of a single-assembly canister. The system comprises a simulated PLUS-7 (16x16) fuel assembly, an internal basket and canister made of Stainless steel 304. To accurately simulate the decay heat of actual spent fuel, 236 electric heater rods are installed within the assembly. The power distribution of these heaters is adjusted to follow a 4-step axial peaking factor, reflecting the non-uniform heat generation profile observed in actual nuclear fuel. Instrumentation includes 96 K-type thermocouples strategically distributed across the fuel rods, basket walls, and the canister surface to capture a comprehensive three-dimensional temperature field. The facility operates under a wide range of internal pressures, from vacuum conditions up to 800 kPa, using backfill gases such as helium and air.



[Fig. 2. Location of installed TCs]

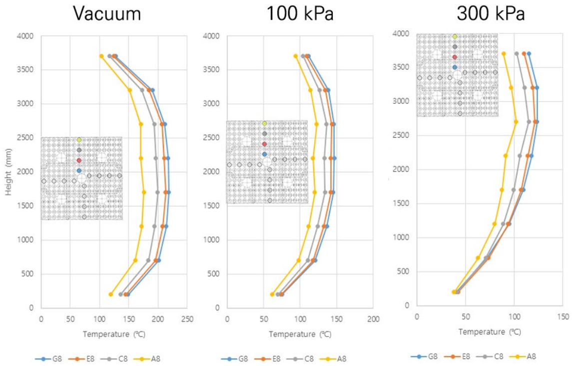
For the numerical analysis, two distinct approaches were employed. The first approach involved three-dimensional CFD analysis using ANSYS Fluent, where the complex geometry of the fuel rods and internal basket was explicitly modeled. A second-order upwind scheme was utilized for spatial discretization, and the Surface-to-Surface (S2S) radiation model was applied to account for radiative heat transfer. The second approach utilized the COBRA-SFS code, a specialized sub-channel analysis tool for SNF storage systems. This dual-track analysis allowed for a comparative study of the codes' predictive capabilities and computational efficiency under various environmental conditions.

3. Results and Discussion

3.1. Analysis of Thermal Characteristics under Variable Conditions

The experimental campaign investigated the influence of decay heat and internal pressure on the thermal profile of the system. As the simulated decay heat increased, a corresponding rise in the overall temperature was observed, accompanied by an intensified temperature gradient between the top and bottom of the assembly. This phenomenon suggests that higher heat loads strengthen the natural convection effects driven by internal gas buoyancy.

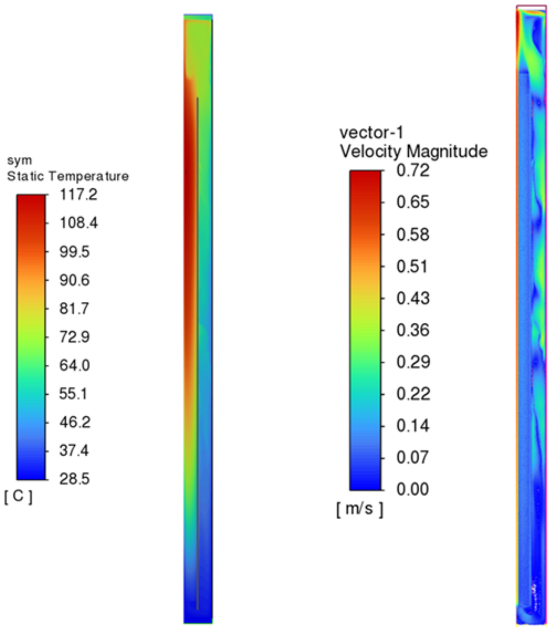
The impact of internal pressure was found to be a dominant factor in heat transfer mechanisms. Under vacuum conditions, heat transfer is restricted primarily to conduction and radiation, resulting in higher average temperatures and a relatively symmetric axial temperature distribution. Conversely, when the canister was pressurized with helium, natural convection became the governing mechanism. This activation of buoyancy-driven flow led to a significant reduction in the PCT and shifted the location of the peak temperature toward the upper region of the assembly. These findings emphasize the necessity of accurately modeling convective flow patterns in best-estimate safety analyses.



[Fig. 3. Axial temperature distribution comparison across different internal pressure]

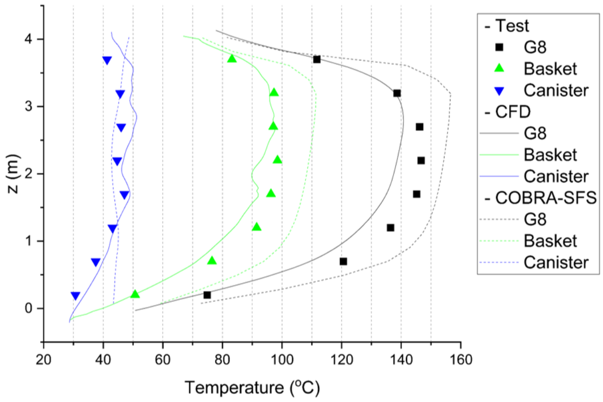
3.2. Validation of Thermal Analysis Models

The numerical results from both CFD and COBRA-SFS were compared against the experimental data obtained from the 96 measurement points. The CFD model demonstrated exceptional accuracy, predicting the PCT with an error margin of less than 5.5%. The simulation successfully captured complex internal circulation, characterized by upward flow within the fuel assembly and recirculating vortices in the annular region between the basket and the canister wall. These vortices, while inducing localized temperature variations, contribute significantly to the overall heat dissipation performance.



[Fig. 4. Axial temperature and velocity profile]

The COBRA-SFS code also exhibited good agreement with the experimental data, showing a maximum PCT error of approximately 13%. Although the sub-channel approach is less detailed than CFD, its computational efficiency proved to be highly effective for rapid sensitivity studies. Both models accurately predicted the shift in PCT location as a function of pressure. Sensitivity analyses further revealed that internal rod emissivity and the axial power profile are the most critical parameters influencing the reliability of temperature predictions.



[Fig. 5. Comparison between measured temperature and predicted temperature using CFD and COBRA-SFS code]

4. Conclusion

This study evaluated the thermal performance of a spent fuel dry storage system using the FACTS facility and validated numerical analysis codes. The experimental results provided clear insights into the role of decay heat and internal pressure in dictating temperature distributions and natural convection formation. The high degree of agreement between the measured data and the CFD predictions (within 5.5%) supports the transition from conservative to best-estimate thermal evaluations. The validated models established in this research will serve as a technical foundation for the design and licensing of domestic dry storage canisters, ultimately enhancing the safety and economic efficiency of spent nuclear fuel management.

5. Acknowledgement

This work was supported by the Institute for Korea Spent Nuclear Fuel (iKSNF) and National Research Foundation of Korea(NRF) grant funded by the Korea government(Ministry of Science and ICT, MSIT)(No. 2021M2E1A1085226)

REFERENCES

[1] J. Benavides et al. “Modeling and Results of the Dry Cask Simulator(DCS) with STAR-CCM+”, Nuc. Eng. Des., vol. 369, 2020

[2] U.S. Nuclear Regulatory Commission, “Standard Review Plan for Spent Fuel Dry Storage Systems at a General License Facility”,NUREG–1536, Rev. 1, 2010.

[3] E.R. Lindgren and S.G. Durbin, “Materials and Dimensions Reference Handbook for the Boiling Water Reactor Dry Cask Simulator”, SAND2019-6079R, Nov. 2017.