Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 12-13, 2016

CFD Analysis on the Passive Heat Removal by Helium and Air in the Canister of Spent Fuel Dry Storage System

Doyoung Shin, Uiju Jeong, Sung Joong Kim*
Department of Nuclear Engineering, Hanyang University
222 Wangsimni-ro, Seongdong-gu, Seoul 04763, Republic of Korea
**Corresponding author: sungjkim@hanyang.ac.kr*

1. Introduction

Use of a spent fuel dry storage system is a practical way to provide long-term (60 years) storage before permanent disposal. Due to the requirement for long-term natural cooling and minimal management, the dry storage system necessitates to adopt passive decay heat cooling mechanism to assure integrity of fuel cladding with ample safety margin.

Adopted passive heat removal process is natural circulation of backfill gas in the canister. Decay heat from the spent fuel assembly drives difference in gravity acceleration and generates buoyancy force, which promotes natural convective heat transfer of backfill gas.

In the current commercial design, the canister of the dry storage system is mainly backfilled with helium gas. Helium gas shows very conductive behavior due to high thermal conductivity and small density change with temperature [1]. However, other gases such as air, argon, or nitrogen are expected to show effective convective behavior. Thus these are also considered as candidates for the backfill gas to provide effective coolability [2].

In this study, to compare the dominant cooling mechanism and effectiveness of cooling between helium gas and air, a computational fluid dynamics (CFD) analysis for the canister of spent fuel dry storage system with backfill gas of helium and air is carried out.

2. Dry Storage System for Interim Storage of the Spent Fuel

Spent fuel keeps generating heat due to decay of fission fragments emitting β and γ particles. Radioactive materials concealed inside the cladding could be released to environment if no appropriate cooling is provided. Therefore, spent fuels should be cooled down in the interim storage system until decay heat is reduced to acceptable and manageable levels.

There are two ways to safely store the spent fuels. One is by putting spent fuels in the water reservoir (wet storage) and the other is dry storage system. Wet storage system is very efficient way to cool down the decay heat. However limitations in storage space and cost bring necessity of adopting a dry storage system. Dry storage system is not very efficient compared to wet storage but minimal management including no need of electrical power is an attractiveness for interim storage of the spent fuels.

2.1 Description of dry storage system

There are various types of spent fuel dry storage system model. TN-24P cask, HI-STORM 100S, etc. are frequently studied in many countries [3, 4]. These dry storage system models are designed to contain 21 to 32 fuel assemblies in one system. Typically, dry storage system consists of stainless steel canister and concrete cask. Figure 1 shows the conceptual design of the HI-STORM 100S. Natural circulation of backfill gas occurs inside the canister and air gap between the canister and the cask.

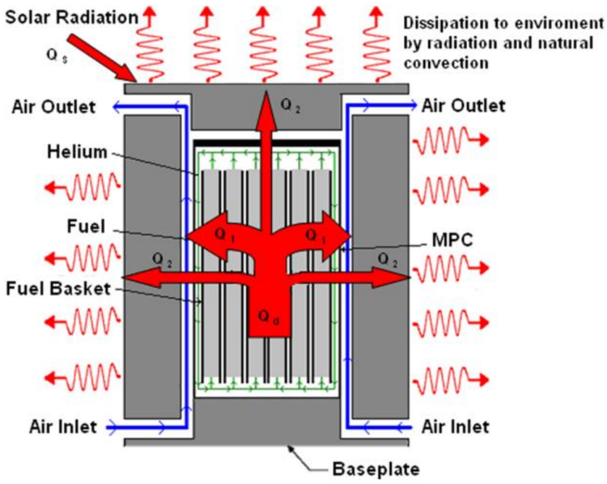

Fig. 1. Conceptual design of HI-STORM 100S [4]

Abovementioned type of dry storage system capable of containing 24 spent fuel assemblies is considered in Korea [5].

3. CFD Modeling

In this study, a commercial CFD analysis tool, namely ANSYS FLUENT 16.2, was employed to simulate buoyancy-induced flow in the canister of dry storage system and to investigate dominant heat removal mechanism depending on its backfill gas. The maximum cladding surface temperature inside the fuel assembly and Reynolds number in each sub-channels are major interests in this stage.

However, modeling full scale of dry storage system is too expensive to simulate. Thus down-scaling was performed. Height was down-scaled by ratio of 1/2 and only one quarter of a 16×16 fuel assembly was considered. Flow area was adjusted by scaling law for single phase natural circulation system [6].

3.1 Modeling of the canister

Figure 2 shows a schematic of the side and the cross-sectional view of modeled geometry, 1/2 height down-scaled canister with 8×8 partial fuel assembly with dimensions. For simplicity, the canister was modified to square shape and only a quarter of 8×8 assembly was modeled assuming symmetry. By the scaling law [6], inner edge length of the canister was determined to 80 mm. Thickness of the canister was conserved. Free air space was modeled to apply convective boundary condition at the outer wall of the canister.

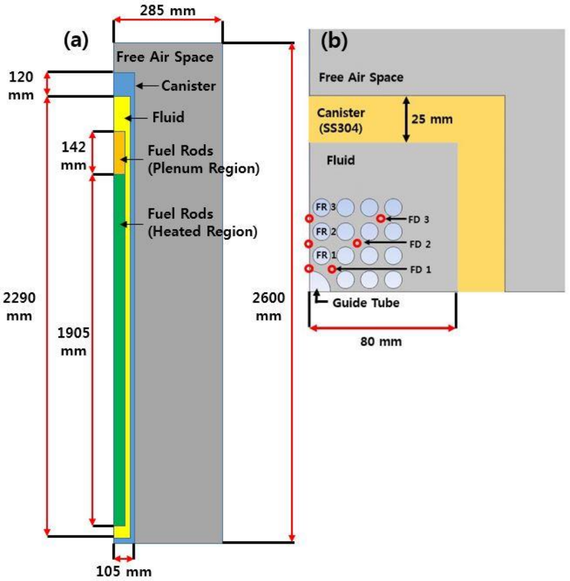

Fig. 2. Schematic diagram (a) side view, (b) cross-sectional view of modeled geometry

3.2 Grid generation

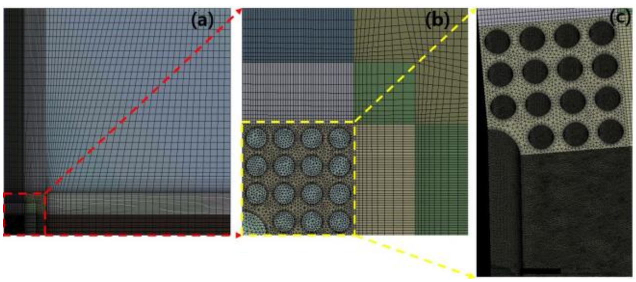

Fig. 3. Generated grids for (a) free air space, (b) canister, (c) fluid region

The modeled section was meshed using ANSYS Mesh program as shown in Figure 3. To reduce the number of high skewed cells generated near the narrow gap between the fuel rods, smaller cell size was utilized. The grid system generated contains about 15,000,000 nodes.

3.3 Solver settings

After the grid generation step, the generated mesh data was imported to FLUENT 16.2. Table I shows information about FLUENT 16.2 solver settings used for the simulation. All explored simulations were calculated under steady and gravitational condition.

Table I. FLUENT solver settings

	Viscous model :	Laminar
	Density model :	Boussinesq
Spatial Discretization	Gradient :	Least Squares Cell Based
	Pressure :	Body force weighted
	Momentum :	1st order upwind
	Energy :	1st order upwind
	Pressure-velocity coupling :	SIMPLE

Moreover, several initial and boundary conditions were considered. Zero velocity and gauge pressure condition was imposed at the inlet and outlet of the free air space to simulate free, unhindered state. A no-slip condition was applied to the fuel rods, guide tube, and walls of the canister. A symmetry condition was used at two side boundary surface. A coupled condition was used at fluid-solid interfaces. Walls of free air space were applied with adiabatic condition. Heat flux condition was used to simulate 6.23 W/rod, which corresponds to 45 MWd/KgU burned-up fuel.

4. Results and Discussion

ANSYS FLUENT enables us to investigate the flow motion of the fluid in the canister by solving the governing equations numerically. All the explored simulations continued until continuity and momentum residuals converged below 1×10^{-3} and that of energy below 1×10^{-7}.

4.1 Helium backfilled canister

Helium gas is most common backfill gas for the dry storage system because of its relatively high thermal conductivity. However, little difference in density with temperature makes helium gas show conductive behavior meaning little natural circulation flow. Figures 4 and 5 show the obtained z-vector velocity distribution in the symmetry plane and axial Reynolds number distribution along FD and FR direction shown in Fig. 2.

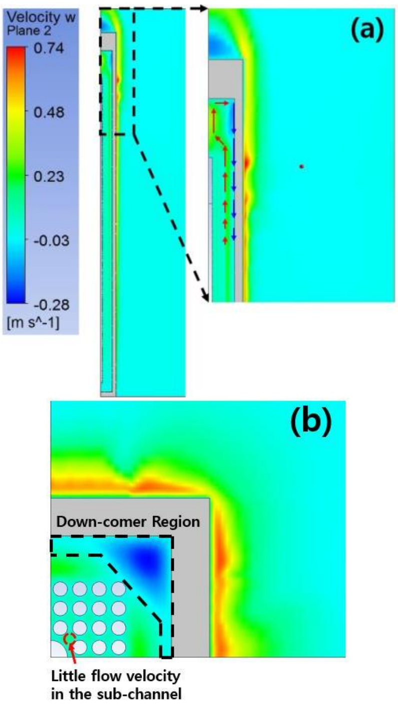

Fig. 4. Z-vector velocity distribution in the (a) symmetry plane, (b) cross-section at interface of heated and plenum region

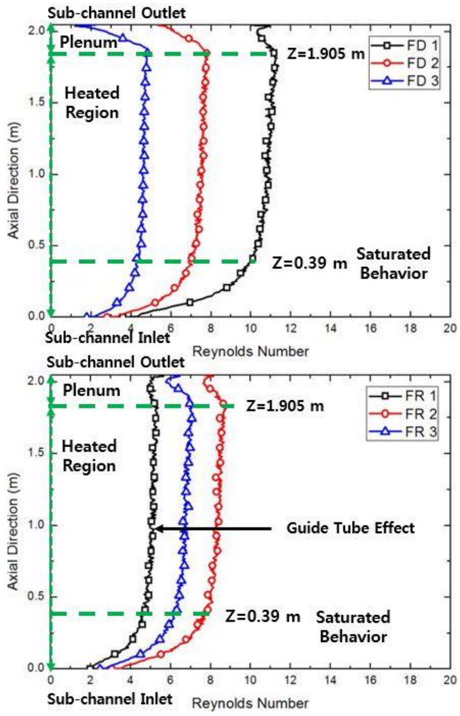

Fig. 5. Axial Reynolds number distribution along FD, FR direction

As shown, natural circulation phenomenon is observed inside the canister. However, weak circulation is observed as Reynolds number range is only about 2~12. Thus the most of heat is removed by conduction.

Figure 6 shows the temperature contour of cladding surface with maximum value of 447.2 K and at the interface of heated and plenum region.

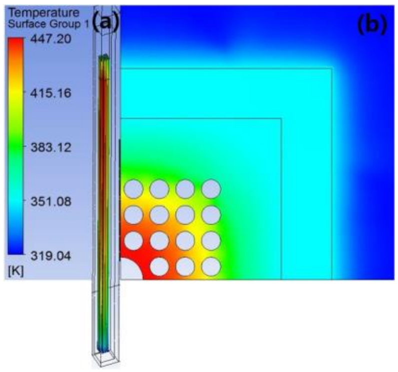

Fig. 6. Temperature distribution at the (a) cladding surface, (b) cross-section at the interface of heated and plenum region

4.2 Air backfilled canister

Unlike helium gas, air shows relatively large density difference with temperature change. Thus, effective natural convection flow is expected. Figures 7 and 8 show the z-vector velocity and axial Reynolds number, respectively for air.

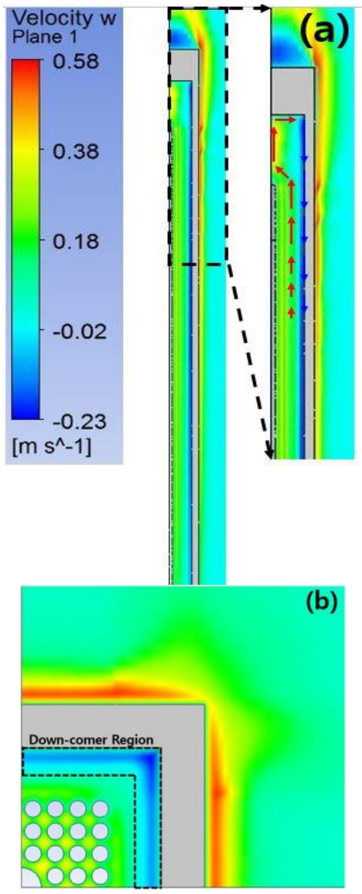

Fig. 7. Z-vector velocity distribution in the (a) symmetry plane, (b) cross-section at interface of heated and plenum region

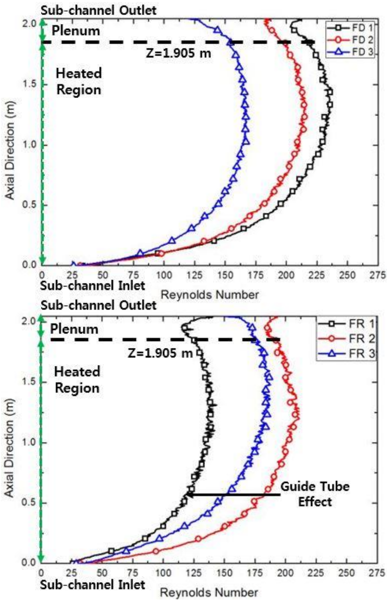

Fig. 8. Axial Reynold number distribution along FD, FR

The stronger natural convection compared to the case of helium is observed inside the sub-channels while flow outside the canister is slower than that of helium. This indicates that less heat is transferred to outer wall of the canister and heat is removed more dominantly by convection. Also Reynolds number ranges 25~240, which is much higher than that of helium.

Figure 9 shows the temperature contour of cladding surface with maximum value of 444.31 K and at the interface of heated and plenum region.

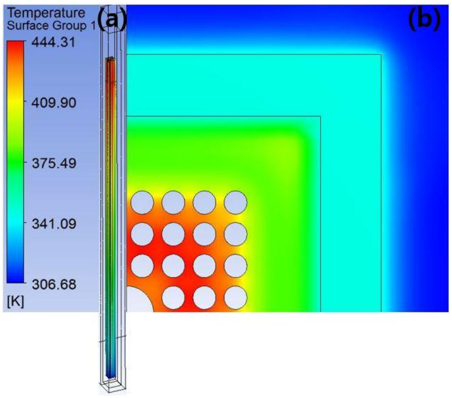

Fig. 9. Temperature distribution at the (a) cladding surface, (b) cross-section at the interface of heated and plenum region

Temperature is uniformly distributed inside the assembly unlike in the case of helium which shows temperature gradient. Also, maximum temperature is located in the plenum region while that of helium is located near the interface of heated region. These observations indicate that in the case of air, heat mixing inside the assembly occurs actively due to convection dominant flow.

5. Concluding Remarks

In this study, CFD simulations for the helium and air backfilled gas for dry storage system canister were carried out using ANSYS FLUENT code. For the comparison work, two backfilled fluids were modeled with same initial and boundary conditions. The observed major difference can be summarized as follows.

■ The simulation results showed the difference in dominant heat removal mechanism. Conduction for helium, and convection for air considering Reynold number distribution.

■ The temperature gradient inside the fuel assembly showed that in case of air, more effective heat mixing occurred compared to helium.

■ There was no large difference in the maximum cladding temperature. However, if simulation was carried out with full assembly scale, the difference is expected to be larger.

For more improved comparison work, an additional simulation is planned including different backfill gases such as argon and nitrogen. Furthermore, using this analysis, natural circulation experiments will be conducted to find structural effect on heat mixing and pressure drop along the sub-channels.

Acknowledgements

This research was supported by the Radioactive Waste Management Technology Program of the Korea Institute of Energy Technology Evaluation and Planning (KETEP), funded by the Ministry of Trade, Industry & Energy, Republic of Korea. (No. 20141710201 66A)

REFERENCES

[1] V. D. Arp, and R. D. McCarty, Thermophysical Properties of Helium-4 from 0.8 to 1500 K with Pressures to 2000 MPa, National Institute of Standards and Technology, 1989

[2] E. W. Lemmon, R. T. Jacobsen, S. G. Penoncello, and D. G. Friend, Thermodynamic Properties of Air and Mixture of Nitrogen, Argon, and Oxygen From 60 to 2000 K at Pressures to 2000 MPa, Journal of Physical and Chemical Reference Data, Vol. 29, pp. 331, 2000.

[3] J. M. Creer, T. E. Michener, M. A. McKinnon, J. E. Tanner, E. R. Gilbert, and R. L. Goodman, The TN-24P PWR Spent-Fuel Storage Cask: Testing and Analyses, Pacific Northwest Laboratory, 1987

[4] L. E. Herranz, J. Penalva, F. Feria, CFD analysis of a cask for spent fuel dry storage: Model fundamentals and sensitivity studies, Annals of Nuclear Energy, Vol. 76, pp. 52-62, 2015.

[5] Korea Radioactive Waste Agency, Technical Report 14220-P1-N-TR-020, 2013

[6] M. Ishii, and I. Kataoka, Scaling Laws for Thermal-Hydraulic System under Single Phase and Two-Phase Natural Circulation, Nuclear Engineering and Design, Vol. 81, pp. 411-425, 1984.