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

Numerical Investigation of Thermal Hydraulic Performance on Variation of Environmental Conditions in Air-Flow Path of Dry Storage Cask

Gyeong-uk, Kang, Dae-sik, Yook, Jung-hoon, Cha, Kwan-hee, Lee, Jung-jin, Kim, Sang-myeon, Ahn
Korea Institute of Nuclear Safety, 62 Gwahak-ro, Yuseong-gu, Daejeon, Korea
gukang@kins.re.kr

1. Introduction

A dry storage cask of the spent fuel has drawn attentions as the saturation of the PWR spent fuel in the wet storage facility of domestic nuclear power plants is expected. A concrete storage cask, one of the dry storage methods, passively removes the decay heat generated from spent fuel assemblies through the air-flow path by buoyancy forces and the outer surface by both conduction and natural convection.

During the normal condition, the thermal performance of the storage cask may be affected by environmental variables such as ambient temperature, wind speed and direction etc.[1]. Thermal evaluation on the storage cask generally assumes a set of fixed environmental factors (e.g., average annual ambient temperature, quiescent conditions, sea level). However, using average values may not be adequate for some sites, as more adverse ambient conditions could exist for prolonged periods of time, allowing a storage system to reach new steady-state conditions that could result in unexpectedly higher spent fuel cladding temperatures as compared to the steady-state conditions analyzed in the safety analysis report for the normal conditions of storage.

The present work numerically investigated the thermal hydraulic performance on various ambient temperatures and wind speeds through air-flow path of the storage cask under normal condition using Fluent 14.5 CFD code [2].

2. Analysis method

2.1 Thermal model configuration and grid

Fig. 1 shows a 3-D 1/4 symmetry finite volume model and the completed grid for thermal analysis, taking into account only the fluid field (air-flow path) and a solid cylinder representing the fuel region as the heat source, and the canister of the storage cask. The air-flow path consists of various geometries with horizontal four air inlets at the bottom and four air outlets at the top, a vertical annulus flow path and circular flow path in the middle. The dimensions for each component used in thermal model refers to the previous work [3]. The grid of the thermal model was constructed with hexahedral and tetrahedral elements, and the total number of the grid is about 1.3 million. As boundary layer is formed in a thin region along the surfaces, the concentrated grids are given near the surfaces of air-flow path using the inflation to improve the simulation accuracy.

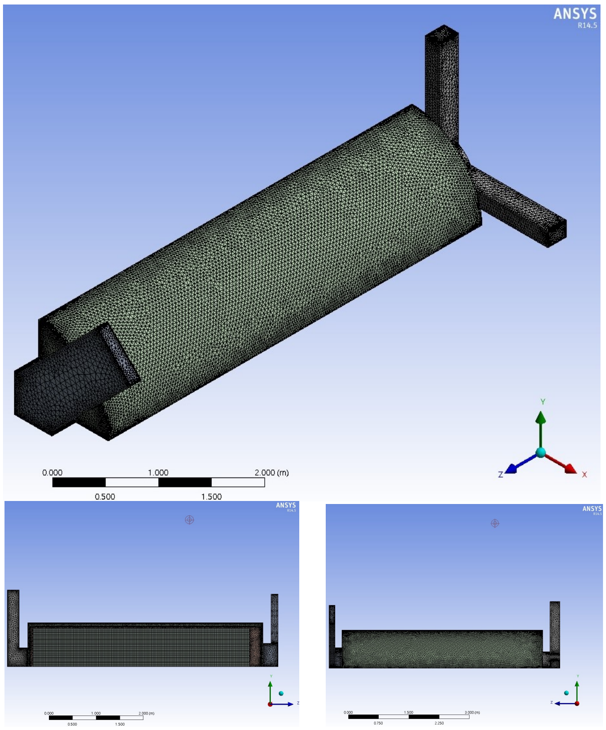

Fig. 1. Finite volume model and completed grid.

2.2 Boundary condition and environmental conditions

Numerical analysis was performed with the Realizable k-ε model for viscous and the Discrete Ordinates (DO) for radiation in Fluent 14.5 CFD code. Incompressible ideal gas was adopted for natural convection heat transfer with air as the working fluid. The total amount of heat generated in the cylinder was assumed at 1,741 W/m^3. The thermal properties for components of storage cask used in this work referred to the previous study [3]. To simulate the heat exchange between outer surfaces of the air-flow path and the external environment, the convective heat transfer coefficient was given to the outer surfaces of the air-flow path. The SIMPLE scheme is used for the Pressure-Velocity Coupling. For pressure discretization, the Body Force Weighted was adopted for high buoyancy forces and the PRESTO! for velocity inlet as the boundary condition at air inlets, respectively.

This study considers two environmental conditions; adverse ambient temperatures ranging from 263.15 (-10 °C) to 573.15 K (300 °C), and wind speeds ranging from 0 to 5 m/s with assuming the wind direction is normal to the air inlets. A total of 30 cases were considered.

3. Results

3.1 Influences on environmental conditions

Fig. 2 representatively shows the entire temperature distribution of the storage cask and velocity distribution of air-flow path for 293.15K and 3 m/s. The flows entering through the inlets rises along the flow path due to buoyance forces and forced flows, and finally escapes to the outlets with cooling the fuel region.

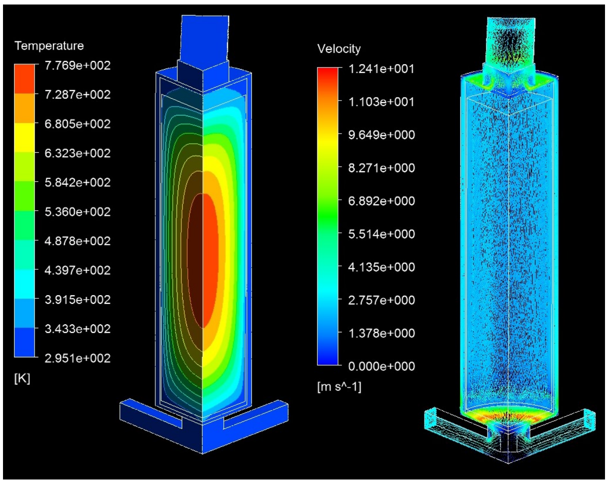

Fig. 2. Entire temperature contour and velocity vector for 293.15K, 3m/s.

Fig. 3 shows the maximum temperatures of fuel region on the various ambient temperatures and wind speeds. It is confirmed that as maximum temperatures increases with higher ambient temperatures, the increase in flow velocity entering into the inlets improved the heat transfer performances by adding the forced flow effects to buoyance forces, resulting in reducing the maximum temperatures.

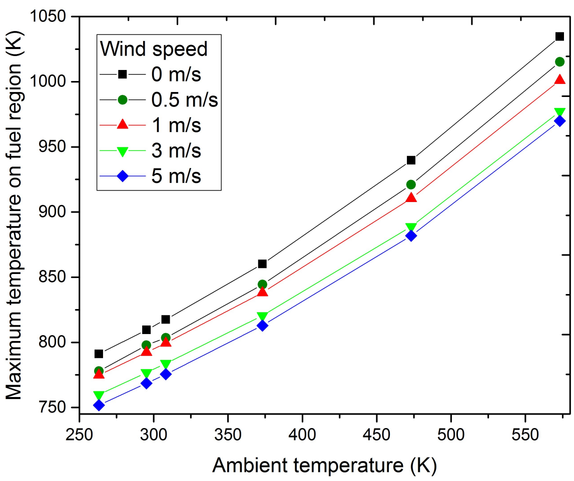

Fig. 3. Maximum temperatures of fuel region on various ambient temperatures.

Mass flow rates measured at air inlets were presented in Table 1. Their values were decreased with increasing the ambient temperatures due to the low temperature differences between hot wall and external environments.

Table 1 Mass flow rates for each case.

Temp.(K)	0m/s	1m/s	3m/s	5m/s
263.15	0.01265	0.03232	0.19317	0.32195
295.15	0.01146	0.02824	0.17223	0.28704
308.15	0.01104	0.02749	0.16496	0.27493
373.15	0.00930	0.02270	0.13622	0.22704
473.15	0.00735	0.01791	0.10743	0.17906
573.15	0.00599	0.01478	0.08869	0.14782

3.2 Investigation on flow fields

As mentioned in Fig. 3, the maximum temperature for components decreases with increasing the wind speeds. To investigate the influences for the magnitude of wind speed, maximum temperature of components was compared with that of each wind speed in Fig. 4. The slopes of all lines from 0 m/s to 1 m/s were steeper than those above 3 m/s. This may be explained that, for wind speeds in the range 0-1 m/s, the flow develops, and wind speeds greater than 3 m/s correspond to the hydrodynamic fully developed-flow, especially in vertical annulus flow path. These phenomenological explanations are clearly shown in Fig. 5.

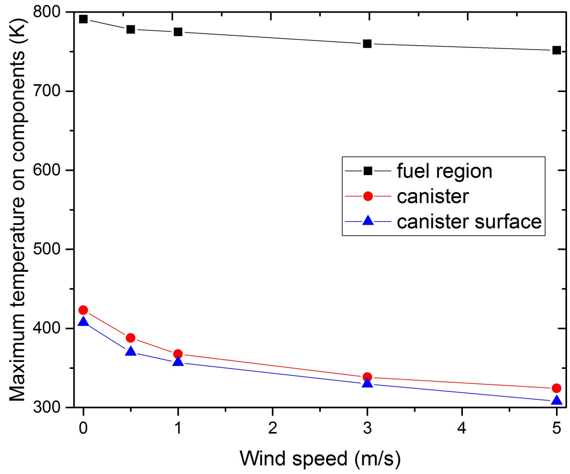

Fig. 4. Maximum temperatures of components on various wind speeds.

Fig. 5 presents the average velocity of each height while the fresh air flows through the horizontal inlet duct, the lower circular, the vertical annulus and upper circular path and the horizontal outlet duct. In the vertical annulus flow path, the velocity is gradually increased in the range of 0-1 m/s, meaning that the velocity profile continues to change with upward flow. However, the velocity is constant regardless of the heights for velocity greater than 3 m/s, meaning that the velocity profiles is not change. The average maximum velocity occurred at horizontal narrow spaces between the canister and flow-path. This reason is due to the size of flow areas and mass flow rates in channel.

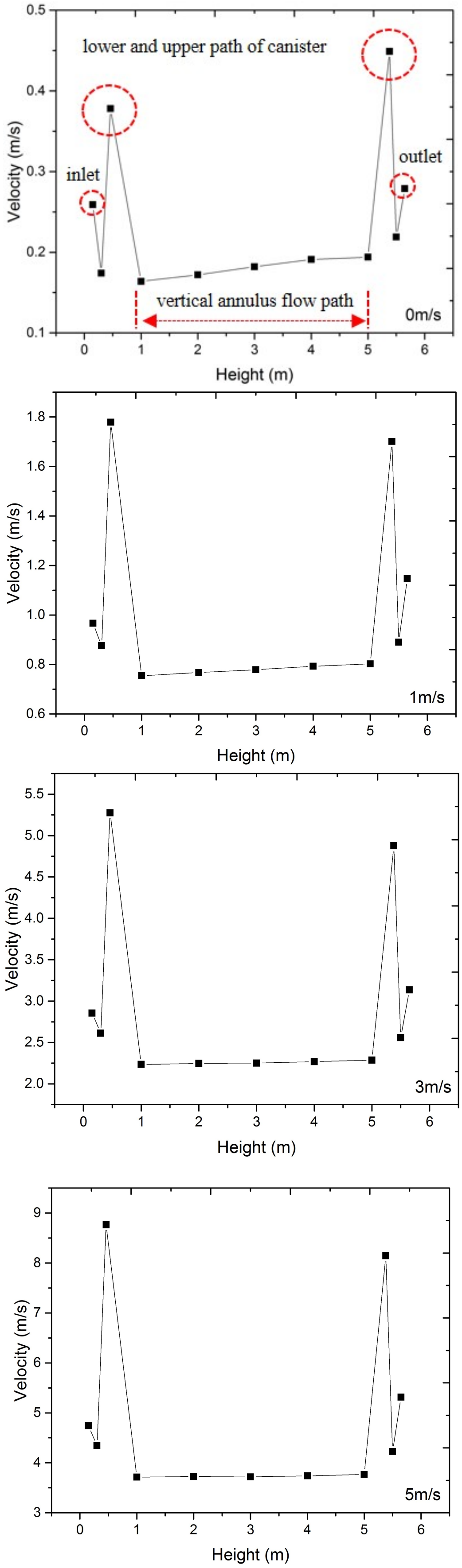

Fig. 5. Average velocity depending on heights of air flow-path for various wind speeds.

4. Conclusion

This study numerically investigated the thermal hydraulic phenomena depending on variation of ambient temperatures and wind speeds in air-flow path of the dry storage cask. The main findings of this study are summarized as follow:

- The maximum temperature increases with higher ambient temperatures and the increase in wind speed improves the heat transfer performances due to adding the forced flows to buoyance forces, resulting in reducing the maximum temperatures.
- In the vertical annulus flow path, the developing flow occurs in the range of 0-1 m/s, the hydrodynamic fully developed-flow occurs for velocity greater than 3 m/s.
- The point where the maximum velocity occurs depends on the size of the flow areas and mass flow rates in channel.

ACKNOWLEDGEMENTS

This work was supported by the Nuclear Safety Research Program through the Korea Radiation Safety Foundation (KORSAFe), granted financial resource from the Nuclear Safety and Security Commission (NSSC), Republic of Korea (NO.1503003).

REFERENCES

[1] U.S NRC, NUREG-2174, Impact of Variation in Environmental Conditions on the Thermal Performance of Dry Storage Cask, 2015.
[2] ANSYS FLUENT User's Guide, release 14.5, ANSYS Inc.
[3] G.U. Kang, H.J. Kim, and C.H. Cho, "Analysis of Flow Fields in Airflow Path of Concrete Dry Storage Cask using Fluent Code", Journal of Computational Fluids Engineering. Vol, No. 2, pp. 47-53, 2016.