A CFD ANALYSIS OF THERMAL BEHAVIOR IN PASSIVE HEAT REMOVAL SYSTEM OF DRY STORAGE CASK UNDER DIFFERENT CONDITIONS.

G.U. KANG, D.S. YOOK, J.H. CHA

Korea Institute of Nuclear Safety, 62 Gwahak-ro, Yuseong-gu, Daejeon, Korea

ABSTRACT

This work numerically investigated the thermal behavior in passive heat removal system of dry storage cask under normal condition and off-normal condition on variation of ambient temperatures and wind speeds using Fluent 14.5. The flow fields in air-flow path depending on the heights were described in detail. These phenomenological explanations will be the background basis for understanding thermal behaviors in passive decay heat removal systems of the dry storage casks.

1. Introduction

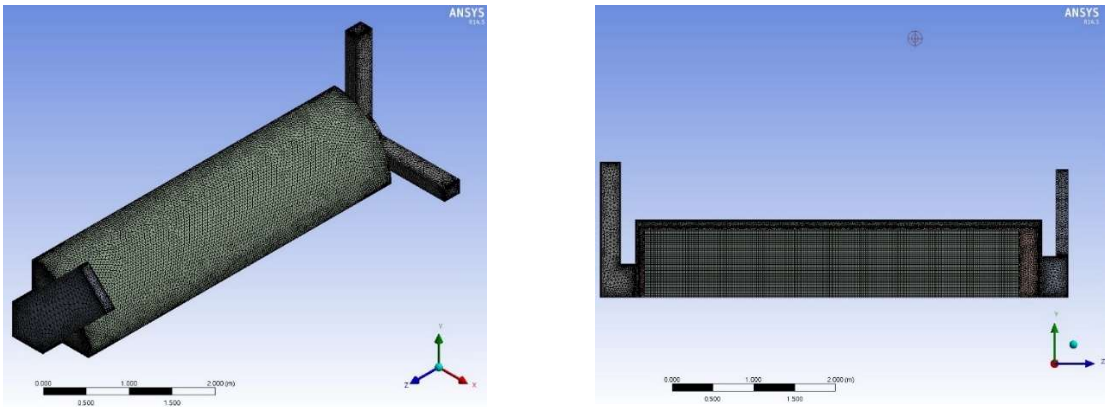
A dry storage cask of the spent fuel has drawn attentions as the saturation of the PWR spent fuel in the wet storage facility of domestic nuclear power plants is expected. A concrete storage cask, one of the dry storage methods, passively removes the decay heat generated from spent fuel assemblies through the air-flow path by buoyancy forces and the outer surface by both conduction and natural convection. The thermal performance on passive decay heat removal system may be influenced by air inlet blockage and diverse external environments such as ambient temperatures, the wind speed and direction during the storage. Thermal analysis 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 [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).

In this study, the thermal analysis is performed to investigate thermal behaviors and flow pattern in the passive decay heat removal system for variations of the decay heat, ambient temperatures and wind speeds under the normal (inlet 100% open, 295.15K), off-normal (inlet partial blockage, 311.15K) 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 acting as canister, i.e., the heat source. The air-flow path consists of various geometries with four horizontal 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 solution 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 developing along the heated wall 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.



(a) Entire view (b) side view

Fig. 1. Finite volume model and completed grid.

2.2 Boundary condition and environmental condition

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. The influences of other viscous models of k-ε and k-ω models on maximum temperatures of each component were investigated and compared. 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) covering the severe cold and fire accident conditions, and wind speeds ranging from 0 to 5 m/s with assuming the wind direction is normal to the air inlets. Under these environmental conditions, thermal analysis was performed for normal condition with air inlet 100% open, and off-normal condition with air inlet 50% blocked.

3. Results

3.1 Comparison of results on normal -and off-normal condition

Table 1 shows the maximum temperatures of fuel region, canister and outer surface of canister calculated from thermal analysis on the adverse ambient temperatures under the normal condition and off-normal condition.

Temp.	295.15(K)		373.15(K)		473.15(K)		573.15(K)	
components	normal	off-normal	normal	off-normal	normal	off-normal	normal	off-normal
Fuel region	809.6	810.6	860.16	861.4	939.73	941.2	1034.6	1036.1
Canister	449.7	451.7	514.1	515.6	598.3	599.4	686.6	687.3
Canister surface	435.2	437.3	500.3	501.8	586.9	588.7	681.7	683.3

Table 1 Maximum temperatures of each component on ambient temperatures.

As ambient temperatures increase, the maximum temperature of each component is higher. With further increases, the temperature differences between canister and canister surface is lower due to the deterioration of heat removal performance by low temperature differences with ambient temperature. Additionally, maximum temperatures on each component under normal condition and off-normal condition are almost similar, meaning that the thermal integrity of the storage cask is secured.

3.2 Flow fields on environmental conditions

Fig. 2 shows the entire temperature distribution and velocity distribution of air-flow path for 293.15K and 0 m/s, representatively. 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. This trend is equal to the results on off-normal conditions. The flow velocity is the fastest in the lower horizontal regions where the flow area is narrow because the mass flow rate is always equal to all elevation of air-flow path.

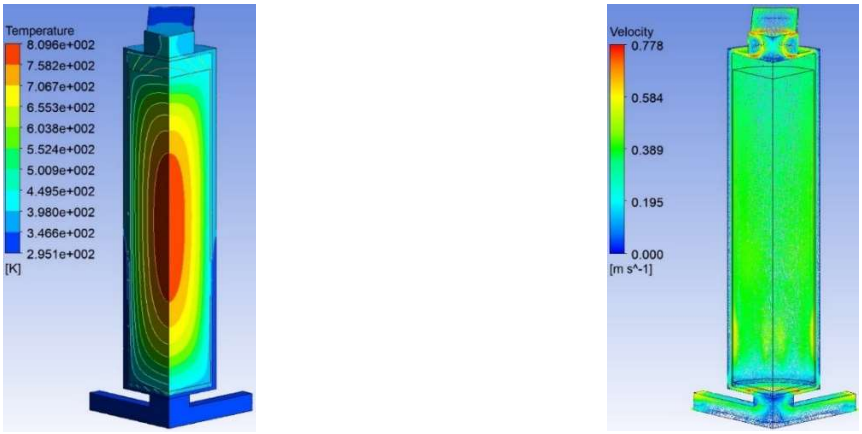
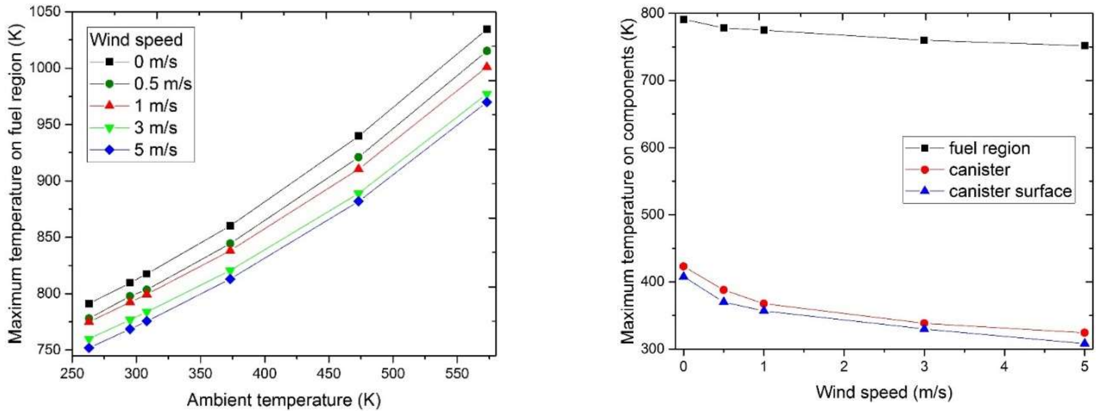


Fig. 2. Entire temperature contour and velocity vector for 293.15K, 0m/s on normal condition.

Fig. 3 shows the results on the environmental conditions. It is confirmed that as maximum temperatures increase with higher ambient temperatures, and 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 in Fig. 3(a).



(a) Max. temperature of fuel region (b) Max. temperature on each component

Fig. 3 Maximum temperature on ambient temperature and wind speed on normal condition.

The maximum temperatures of each component was compared with those on each wind speed in Fig. 3(b). 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. 4. Fig. 4 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 velocities 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.

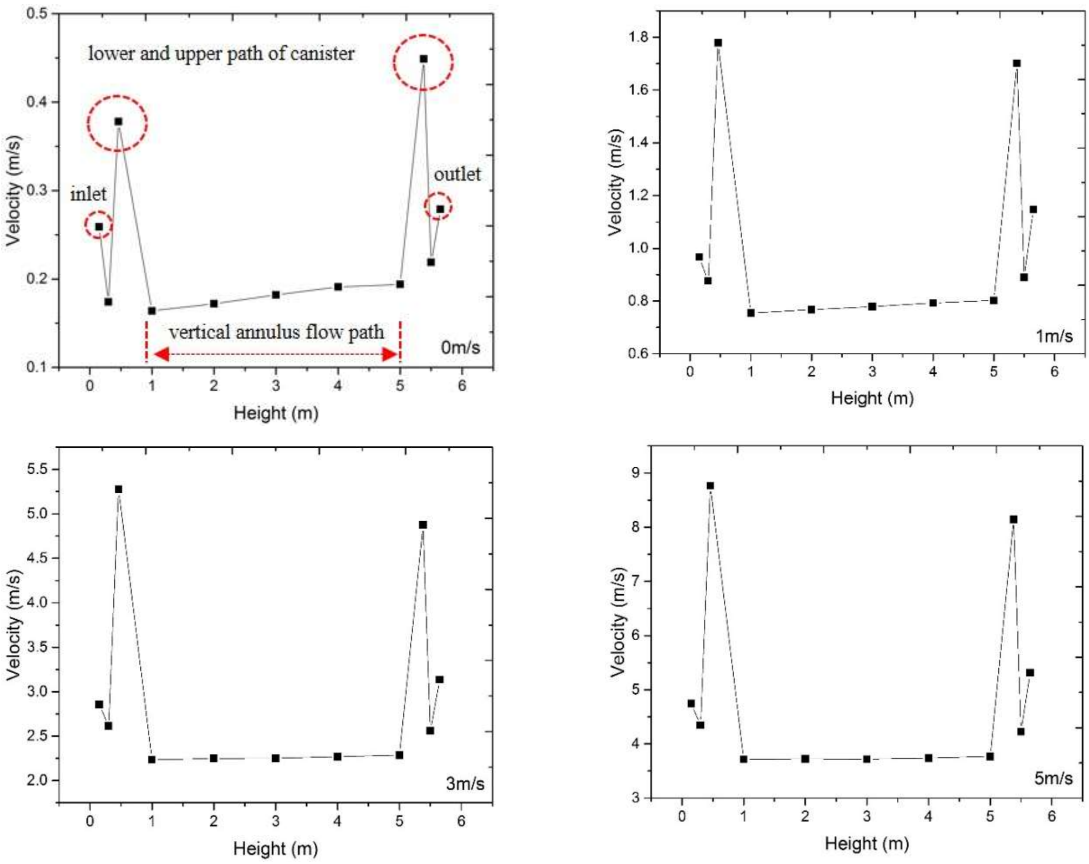
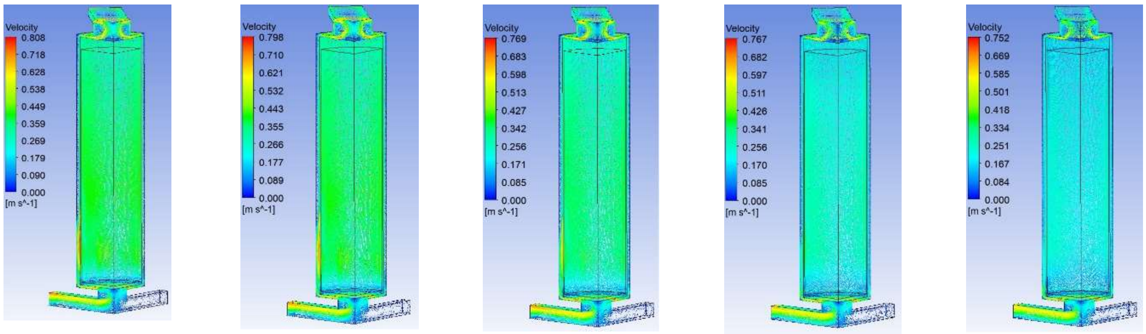


Fig. 4. Average velocity depending on heights of air flow-path for wind speeds on normal condition.

Fig. 5 show the flow fields on ambient temperatures ranging from 273.15 K to 573.15K under the off-normal condition. Since the air inlet on one side is blocked, outside fresh air is entered into the other side by buoyancy forces. The maximum velocity occurs in the horizontal region near the bottom as mentioned above, and as the ambient temperature rises, the maximum velocity in this region decreases due to lower temperature differences, implying the deterioration of heat removal performance.



273.15K 308.15K 373.15K 473.15K 573.15K

Fig. 5. Vector on ambient temperatures on off-normal condition.

The horizontal region near the bottom serves to distribute the flow entering into air inlets uniformly. However, as shown in Fig. 5, it is found that the flow distribution was not uniform in a vertical annular region with a low height due to blockage of one side of the air inlets. It may reduce the local heat transfer performance of the canister. Flow behaviors at cross-section of air-flow path on flow direction is shown in Fig. 6 and 7, respectively. Fig. 6 shows the flow fields depending on the heights.

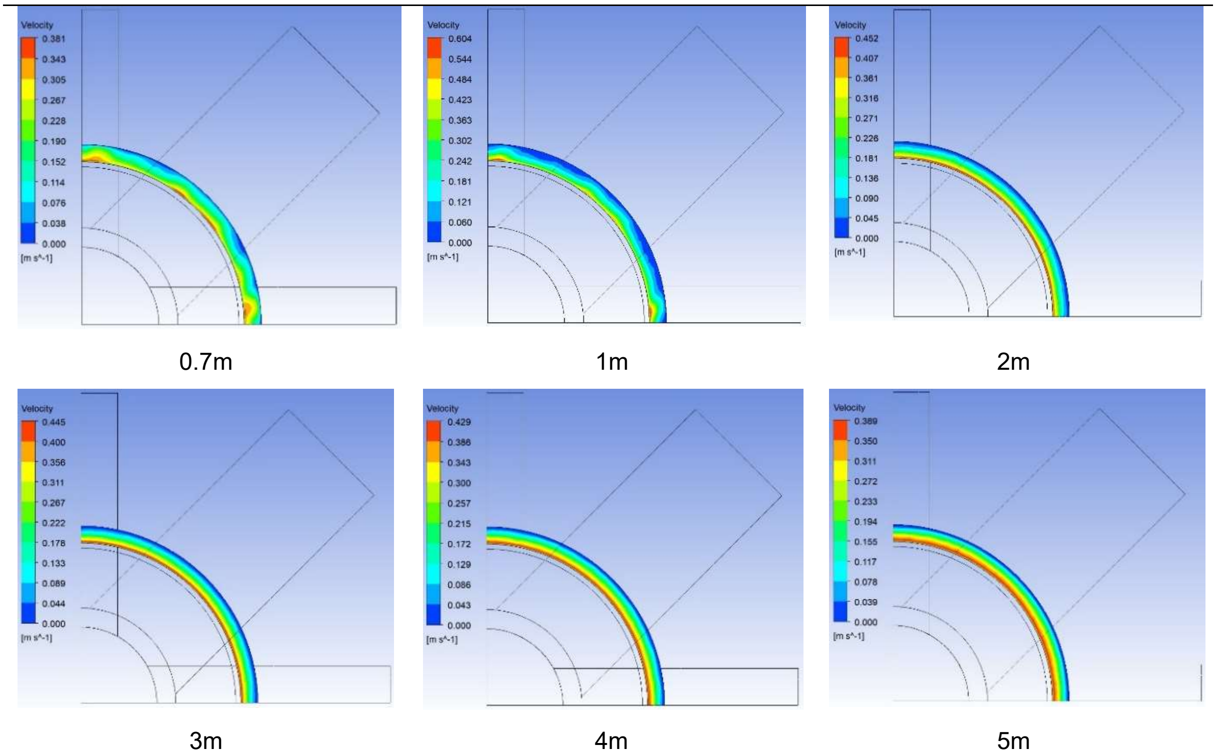


Fig. 6. Velocity contour of cross-section in air-flow path on each height under normal condition.

The flows entering into the air inlets pass through annular flow-path with forming the plumes from flow region near the bottom. At heights more than 2 m, the flumes occurring from the bottom gradually merged with neighboring plumes and the merged flows develop along the walls.

Fig. 7 shows the flow fields on off-normal condition. Plume occurs at the bottom region only in one region because the one side is blocked. However, as the flow develops, the plume merges with the neighboring flows and finally, the flow behavior becomes the same to that of Fig. 6.

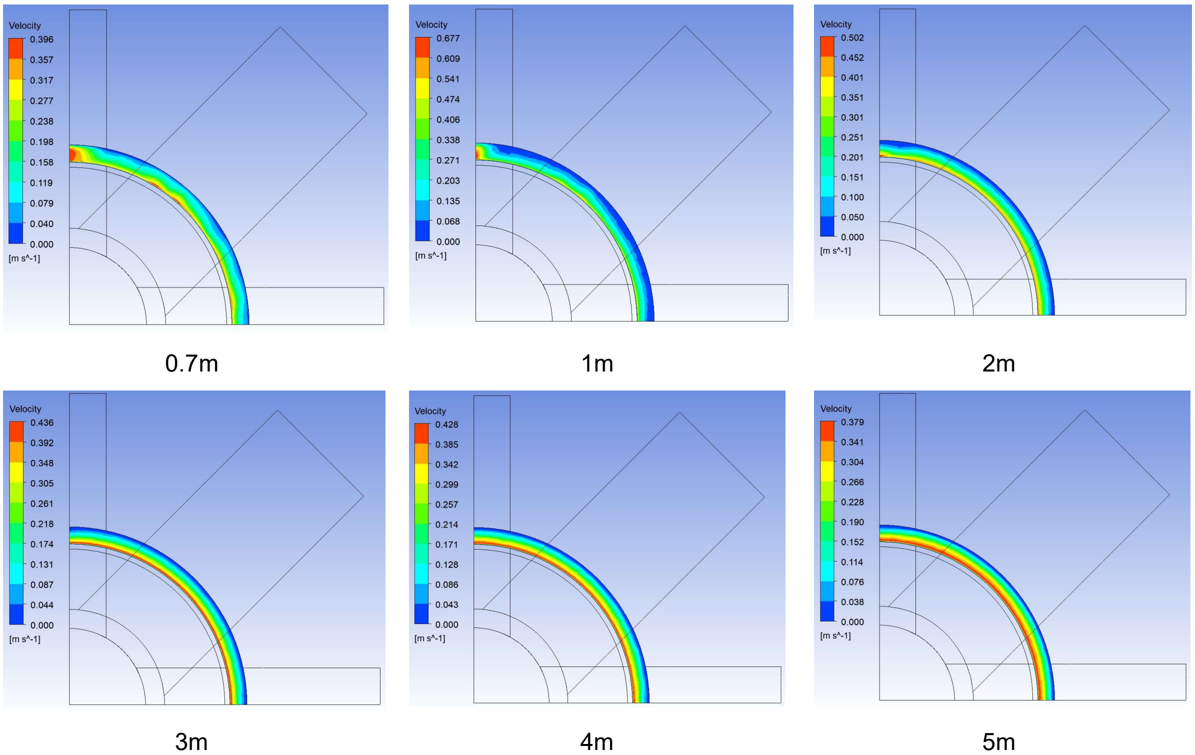


Fig. 7. Velocity contour of cross-section in air-flow path on each height under off-normal condition.

4. Conclusion

This study numerically investigated the thermal behavior in passive heat removal system of dry storage cask under normal condition and off normal condition depending on variation of ambient temperatures and wind speeds. 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. The maximum temperatures of each component at two conditions are almost similar, meaning that the thermal integrity of the storage cask is secured. The temperature and velocity distributions in air-flow path on the heights were compared in detail on each condition. These phenomenological explanations will be the background basis for understanding thermal behaviors in passive decay heat removal systems of the dry storage casks.

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. 1803015)

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.