Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 29-30, 2014

CFD Simulation of Heat and Fluid Flow for Spent Fuel in a Dry Storage

Wang-Kee In [a*], Young-Kyun Kwack [a], Dong-Hak Kook [a], Yang-Hyun Koo [a]
[a]*LWR Fuel Development, Korea Atomic Energy Research Institute, Daejeon, Korea 305-353*
[*]*Corresponding author: wkin@kaeri.re.kr*

1. Introduction

A dry storage system is used for the interim storage of spent fuel prior to permanent depository and/or recycling. The spent fuel is initially stored in a water pool for more than 5 years at least after dispatch from the reactor core and is transported to dry storage. The dry cask contains a multiple number of spent fuel assemblies, which are cooled down in the spent fuel pool. The dry cask is usually filled up with helium gas for increasing the heat transfer to the environment outside the cask. The dry storage system has been used for more than a decade in United States of America (USA) and the European Union (EU). Korea is also developing a dry storage system since its spent fuel pool is anticipated to be full within 10 years. The spent fuel will be stored in a dry cask for more than 40 years.

The integrity and safety of spent fuel are important for long-term dry storage. The long-term storage will experience the degradation of spent fuel such as the embrittlement of fuel cladding, thermal creep and hydride reorientation. High burn-up fuel may expedite the material degradation. It is known that the cladding temperature has a strong influence on the material degradation. Hence, it is necessary to accurately predict the local distribution of the cladding temperature using the Computational Fluid Dynamics (CFD) approach [1-4].

The objective of this study is to apply the CFD method for predicting the three-dimensional distribution of fuel temperature in a dry cask. This CFD study simulated the dry cask for containing the 21 fuel assemblies under development in Korea. This paper presents the fluid velocity and temperature distribution as well as the fuel temperature.

2. CFD Models and Analysis

A dry cask for the localization (Fig. 1) was designed to store 21 spent fuel assemblies. The cask (canister assembly) consists of 21 baskets and 22 spacer disks. The basket is a rectangular steel housing to contain a single fuel assembly. Three types of spacer disks are used to support the baskets.

This study used a two-step CFD approach for predicting the heat and fluid flow in the cask. The first approach models the dry cask by assuming the fuel assemblies in the baskets as a porous body. This cask model predicts the helium flow and temperature in the baskets as well as the canister. The second approach simulates the heat and fluid flow in a single fuel assembly within the basket. The first CFD calculation gives the inlet boundary conditions (flow velocity and temperature of helium) for the second CFD simulation.

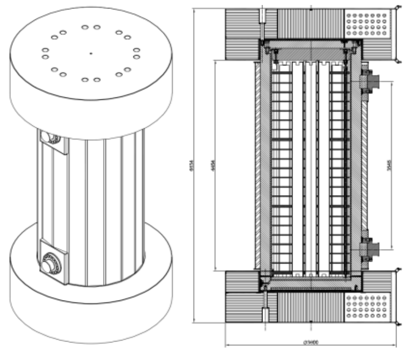

Fig. 1. Dry cask for transportation and storage of spent fuel.

2.1 Dry Cask Model

A dry cask model (Fig. 2) was developed using a porous body assumption of the fuel assembly. The cask model includes the canister (steel vessel), 21 baskets and porous bodies of the fuel assemblies, and 22 spacer disks. The diameter and height of the canister are 1630 mm and 4580 mm, respectively. The basket is a steel duct with a cross section of 222 x 222 mm. The thickness and height of the basket are 5 mm and 4550 mm, respectively. The distance between the baskets is 30 mm. The porosity (ratio of fluid volume to total physical volume) of the fuel assembly is 0.56, and the active heated length is 3800 mm. The spacer disks consist of 4 main disks (height 50 mm, span 1409 mm) and 18 intermediate disks (height 20 mm, span 197 mm).

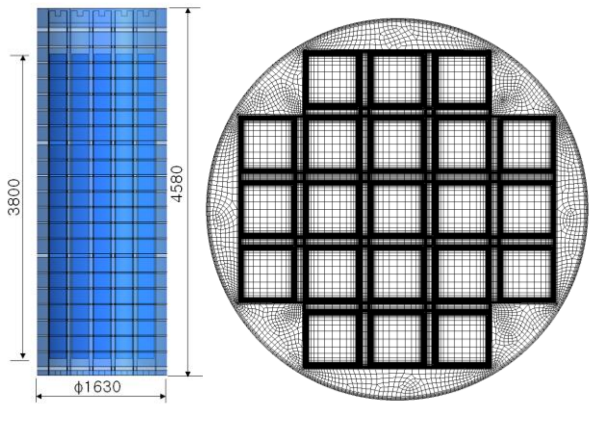

Fig. 2. Dry cask model and cross-sectional mesh.

The hexahedral mesh is generated in the basket region using a multi-block technique. The quadratic mesh is used in the peripheral region of the cask. The cask model consists of approximately 18 million cells and 20 million nodes.

A constant heat source is given in the active fuel region to simulate the decay heat of spent fuel. A constant temperature is set at the side wall of the canister and adiabatic condition at the bottom and top walls of canister. The pressure loss coefficient for the porous fuel body is assumed to be 25 in the axial direction and 2500 in the transverse direction.

2.2 Fuel Assembly Model

A fuel assembly model (Fig. 3) was created to model the detailed structure of the fuel assembly in a basket. This CFD model includes a single fuel assembly and basket. The fuel assembly for this CFD study is a 16x16 square rod bundle with 11 spacer grids for use in a Korean optimum power reactor, the OPR1000. The pitch-to-diameter ratio of a rod bundle (P/D) is 1.35 with a rod diameter (D) of 9.5 mm. The total length of the fuel rod is 4082 mm, and the heated length is 3800 mm. The spacer grid consists of 9 mid-grids with a mixing vane and bottom and top end grids. The height and span of the grid are 40 mm and 400 mm, respectively. This CFD model also includes the rod support structures of the spacer grid such as the spring and dimple.

A special meshing technique was applied to generate trimmed hexahedral cells for the complicated fuel assembly. The three prism layers are created on the surface of the fuel rod and guide tube in order to more accurately simulate the thermal and fluid condition near the wall. The total number of meshes is approximately 190 million cells.

A constant heat flux is given on the surface of the fuel rod and adiabatic condition in the guide tube. The side wall of the basket was assumed to be adiabatic for conservatism. The inlet boundary condition of the helium velocity and temperature is imported from the cask model result. A constant pressure is applied to the outlet boundary. No slip condition was set to the wall boundaries. The inlet and outlet boundaries are defined as the cross sections of the basket upstream and exit of the fuel assembly.

2.3 CFD Analysis

The commercial CFD codes ANSYS CFX v14.5 [5] and Star-CCM+ v8.04 [6] were used to perform the CFD analysis for the cask model and fuel assembly model, respectively. The decay heat of spent fuel is set to 1.0 kW per fuel assembly by assuming the storage in the water pool for approximately 5 years. The full buoyancy model was applied to simulate the natural convective flow driven by the decay heat from the spent fuel. A coupled solver was used to obtain a fast and stable solution through an iteration procedure.

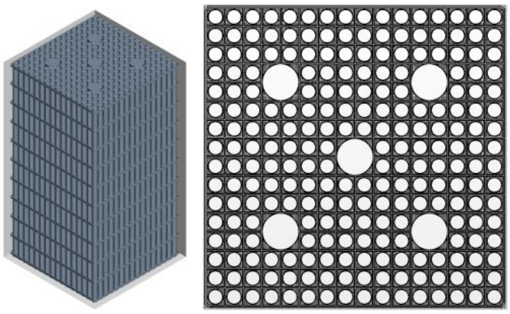

Fig. 3. Fuel assembly model and cross-sectional mesh.

The working fluid in the dry storage cask is helium at 1 bar with a reference temperature of 320 K for the buoyance calculation. The helium flow was assumed to be laminar because the Rayleigh number (Ra) is estimated to be on the order of 10^7. The critical Ra number for a turbulent flow is known to be 10^9. The heat transfer by radiation was neglected for a conservative prediction of the fuel temperature in this study.

The CFD analysis for the cask model used a constant heat source of 6.2 kW/m^3 in a porous fuel body, which is equivalent to 1.0 kW of a single fuel assembly. A constant temperature of 320 K is set at the side wall of the cask, while an adiabatic condition is used at the bottom and top walls of the cask.

The CFD analysis for the fuel assembly model used a constant heat flux of 37 W/m^2 in the active region of the fuel rod. The heat flux distribution is assumed to be uniform in the axial and radial directions. The helium velocity and temperature at the inlet boundary were taken from the CFD result for the cask model. A relative pressure of 0 Pa was set for the outlet boundary.

3. Numerical Results

The CFD analysis of the cask model predicted the overall fluid flow and heat transfer in the dry storage. Fig. 4 shows the helium flow velocity and temperature distributions in the vertical center plane. The helium flow is generated in the upward direction driven by the decay heat of spent fuel inside the basket. The helium velocity is continually accelerated as the helium moves upward. The helium flow is directed to the peripheral region of the cask in the basket outlet. The helium flows downward through the peripheral region and the outer region of the basket. Then, the helium enters each of the baskets in the lower section of the cask. The helium heats up inside the basket as it goes upward and cools down in the peripheral region. The helium temperature in the center basket is higher than that in the other baskets. The temperature rise in the basket appears to be more than 90 ^oC.

Fig. 5 shows the helium velocity and temperature distributions at the top section of the spent fuel. The helium velocity and temperature are uniformly distributed in the center basket but unevenly distributed in the outer baskets. The bulk helium velocity is approximately 0.32 m/sec in all of the baskets while the entrance velocity is 0.19 m/sec in the center basket and 0.09 m/sec in the outer basket. The buoyance effect increased the helium velocity. The mean helium temperature is 266 ^oC in the center basket and 200 ^oC in the outer basket, respectively. The temperature rise can be estimated to be 200 ^oC in the center basket and 90 ^oC in the outer basket.

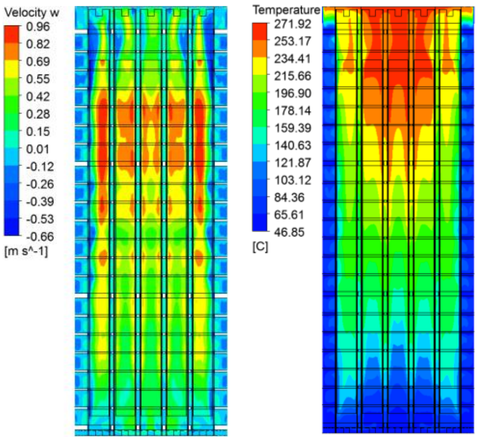

Fig. 4. Velocity and temperature distributions in the cask.

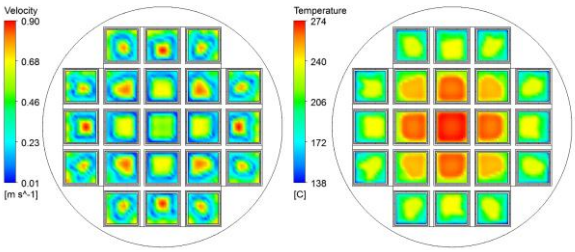

Fig. 5. Velocity and temperature distributions in the baskets at the top section of porous spent fuel.

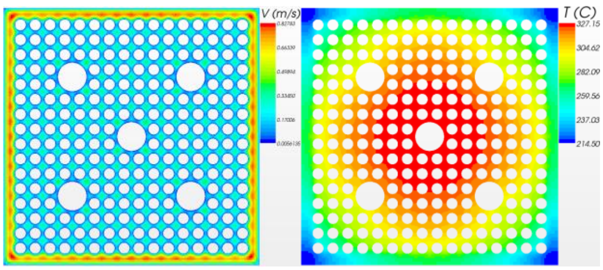

Fig. 6. Velocity and temperature distributions at the heated end of spent fuel assembly in center basket.

Fig. 6 shows a cross sectional view of the helium velocity and temperature at the heated end of the spent fuel assembly, which is stored in the center basket. The large helium flow occurs in the thimble subchannels surrounding the guide tubes and in the side/corner subchannels along the basket wall. This is because the flow area in those subchannels is larger than the matrix subchannels between the fuel rods. The helium temperature in the central region is higher than the temperature in outer region. The maximum temperature of the fuel rod is approximately 330 ^oC in the central spent fuel.

4. Conclusions

A two-step CFD approach was applied to simulate the heat and fluid flow in a dry storage of 21 spent fuel assemblies. The first CFD analysis predicted the helium flow and temperature in a dry cask by a assuming porous body of the spent fuel. The second CFD analysis was to simulate a spent fuel assembly in the single basket by using the entrance boundary condition from the first cask analysis. The CFD results can be summarized as follows.

(1) An upward fluid (helium) flow was induced by decay heat inside the basket and a downward flow in the peripheral region of the cask by cooling.
(2) The helium velocity was accelerated to approximately 0.32 m/sec at the basket exit due to the buoyance effect being more significant in the outer basket.
(3) The helium temperature rise was predicted to be 200 ^oC in the center basket and 90 ^oC in the outer basket, respectively.
(4) The helium and fuel temperature in the central region of the fuel assembly is higher than the temperature in the outer region. The maximum fuel temperature in the center basket is estimated to be 330 ^oC.
(5) The radiation heat transfer should be included in the future CFD analysis for the best estimate of the spent fuel temperature.

Acknowledgements

This work was supported by the National Research Foundation of Korea (NRF) funded by the Korea government (MSIP) (NRF-2012M2A8A5025824).

REFERENCES

[1] S. R. Suffield, J. M. Cutta, J. A. Fort, B. A. Collins, H. E. Adkins, and E. R. Siciliano, Thermal Modeling of NUHOMS HSM-15 and HSM-1 Storage Modules at Calvert Cliffs Nuclear Power Station ISFSI, PNNL-21788, October 2012.
[2] R. A. Brewster, E. Baglietto, E. Volpenhein, and C. S. Bajwa, CFD Analysis of the TN-24P PWR Spent Fuel Storage Cask, PVP2012-78491, Proceedings of the ASME 2012 Pressure Vessels & Piping Division Conference, July 15-19, 2012, Toronto, Canada.
[3] W. K. In, C. H. Shin, T. H. Chun, K. W Song, and J. W. Choi, Computational Fluid Dynamics Analysis of Thermal Flow in Test Facility for Spent Fuel in Dry Storage (in Korean), Proceedings of Korean Radioactive Waste Society, Autumn Meeting, October 7-8, 2010, Busan, Korea.

[4] Y. K. Kwack, W. K. In, C. H. Shin, T. H. Chun, and D. H. Kook, CFD Simulation of Spent Fuel in a Dry Storage, Proceedings of Korean Nuclear Society, Autumn Meeting, October 24-25, 2013, Gyeongju, Korea.
[5] ANSYS CFX v14.5, User Guide, ANSYS Inc., 2013.
[6] STAR-CCM+ v8.04, User Guide, CD-adapco, 2013.