Transactions of the Korean Nuclear Society Spring Meeting
Pyeongchang, Korea, May 27-28, 2010

Full-scale Simulation of a Dry Storage Cask by Computational Fluid Dynamics

Seung Hun Yoo, Hyeun Min Kim, Hee Cheon NO[*]
Korea Advanced Institute of Science and Technology,
335 Gwahak-ro, Yuseong-gu, Daejeon, Republic of Korea,
yoosh@kaist.ac.kr, jeannette@kaist.edu,hcno@kaist.ac.kr[*]

1. Introduction

The prediction of flow and heat transfer in the dry storage system has been carried out to meet the need for thermal-hydraulic analysis for cask designs or to investigate local and global phenomena which could not be measured by the experiments. Computational fluid dynamics (CFD) codes have been used for solving heat transfer inside the complex geometries such as spent fuel assemblies. And more accurate calculations were performed by them compared with those of one-dimensional system codes. In order to provide for the reliability and the guidance of the use of CFD codes, diverse researches were performed with different dry systems and codes. In the present study, TN24P cask was selected for the reference experiment [1]. FLUENT code with a set of CFD models was used for the full-scale simulation of TN24P cask and its results were compared with the experimental data and COBRA-SFS results.

2. Description of TN24P cask

2.1 Test apparatus

A schematic of the TN24P cask was represented in Fig. 1. The thermal-hydraulic test was performed by three different backfills; nitrogen, vacuum and helium, and two different orientations; vertical and horizontal cask orientations. Among six runs, the helium backfilled and vertically oriented experiment was selected for benchmark for FLUENT simulation.

2.2 COBRA-SFS code analysis

COBRA-SFS code was developed and optimized for thermal-hydraulic analysis of dry storage system, which is a steady-state, lumped-parameter, finite-difference computer code [1]. In the simulation of TN24P cask, empirical relationships were used to close the set of equations. The surface friction for all of the subchannels was approximated using a friction factor of f=100/Re, which was derived for a square rod array with pitch-to-diameter ratios typical of PWRs [2]. And the convective heat transfer from the rod and wall surfaces to the fluid was approximated using a film coefficient having the form Nu=3.66 [3]. In the radiation model, The two-dimensional exchange factors for the fuel rods and walls were calculated using one-quarter pin surface segments and the cross-string correlation method of Hottel [4].

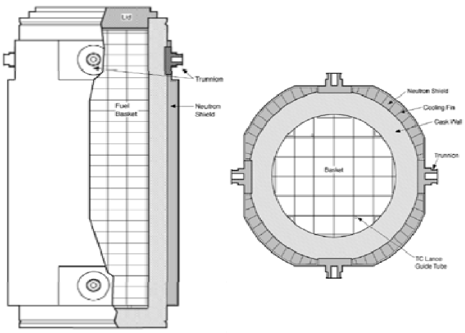

Fig. 1. Top and side views of TN24P Cask (Creer et al., 1987).

3. Full-scale simulation of TN24P cask

3.1 Mesh generation

The grid of TN24P cask was generated by GAMBIT [5]. The two-dimensional mesh on the cross section of TN24P cask was generated and extruded along axial direction to consist of the three-dimensional mesh. By assuming that the helium flow would be symmetric, the numerical domain was simulated by its 1/8section. A three-dimensional mesh was generated, which is approximately 4.7 million cells (Fig. 2).

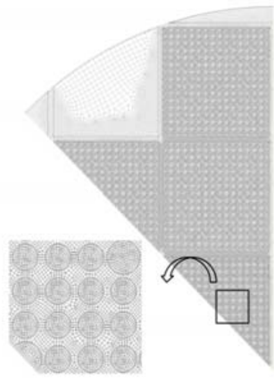

Fig. 2. Mesh generation for 1/8 section of TN24P cask.

3.2 Numerical models

Flow regime inside cask was modeled by laminar model and SIMPLE algorithm was selected for pressure and velocity coupling. For radiation heat transfer, Discrete Ordinates radiation model was used with 4 x 4 discretized divisions along theta and phi angle. The double precision was selected to prevent a possible round-off error in the complex geometries of fuel assemblies.

3.3 Boundary conditions

Fuel rods inside assemblies were modeled by volumetric heat source. And the predicted decay heat profile by gamma measurement was applied. The symmetry condition was applied for numerically cut surfaces in the 1/8 section. The basket was modeled by considering gap sizes between basket and canister inner wall and between basket plates.

3.4 Numerical results and discussions

Full-scale simulation was performed by 6 CPUs of Intel Xeon 2.66 GHz with a Windows XP 64bit operating system. Fluent calculation was compared with experiment and COBRA-SFS simulation in Table 1. The guide tube temperature in location of GD1 and estimated PCT were agreed within 1°C. The licensing temperature of fuel cladding is 380°C, so 1°C is regarded as negligible temperature. Fig. 3 shows temperature distributions in spent fuel assemblies and basket.

Table 1: Measured and estimated temperatures in TN24P cask.

Case	Meas. Temp. (D1)	Estimated PCT
Experiment	214.0 °C	-
COBRA-SFS	-	221.0 °C
FLUENT	214.8 °C	220.5 °C

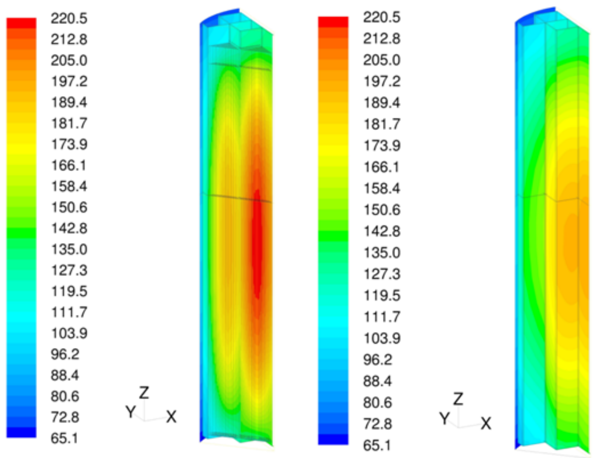

Fig. 3. Temperature distribution in spent fuel assemblies (left) and basket (right).

3. Conclusions

Full-scale simulation of TN24P cask was performed by FLUENT code. The spent fuel assembly was modeled by explicit manner. The simulation results with FLUENT code showed the good agreement with the experimental data and COBRA-SFS results. FLUENT code was validated with a set of CFD models. And the feasibility of the proposed analysis method for full-scale dry storage cask was confirmed.

REFERENCES

[1] Creer, J.M., et al., The TN-24P Spent Fuel Storage Cask: Testing and Analysis, Electric Power Research Institute, NP-5128 ,1987
[2] E.E. Sparrow and A.L. Loeffler, Jr., Longitudinal Laminar Flow Between Cylinders Arranged in Regular Array, AIChE journal, Vol.5(3), p.325-330, 1959
[3] W.M. Kays and M.E. Crawford, Convection Heat and Mass Transfer, McGraw-Fill, 1980
[4] R.L. Cox, Radiation Heat Transfer in Arrays of Parallel Cylinders, Oak Ridge National Laboratory, ORNL-5239, 1977
[5] FLUENT Inc., FLUENT 6 User’s Guide, 2003