Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 22, 2009

Validation of CFD Methods Using Measured Data For Horizontal Concrete Storage Module

Dong-Gyu Lee [a*], Jea-Ho Park [a], Kyu-Sup Choi [a]
[a]*Korea Nuclear Engineering & Service Co., #6F, Hapdong Bldg., 210-2 Yangae-dong, Seocho-gu, Seoul, Korea*

1. Introduction

Computational fluid dynamics (CFD) methods are currently being used to design and analyze spent fuel storage and transportation casks. To gain a better understanding of CFD's capabilities to analyze the complex heat transfer and flow phenomena occurring in a passive dry storage system, CFD model needs to be developed correctly and validated using test data. Additionally CFD model must be validated to minimize modeling and application uncertainties [1]. In particular, the turbulence modeling of buoyancy driven flow can greatly influence the final results if not applied correctly. Therefore, to reduce uncertainties for turbulence modeling, different turbulence models are required to be compared to the test data.

This paper addresses the validation of CFD method using measured temperature data obtained from HCSM (Horizontal Concrete Storage Module) under long term storage conditions.

2. Thermal Test Model and Analysis

In this section, the test model and methods applied to the validation of HCSM are described.

2.1 Thermal Test Model and Test Condition

The test model for HCSM consists of a reinforced concrete module and a stainless steel canister enclosing 24 baskets, which contains spent nuclear fuel assembly (SNF). The structural support for baskets inside of the canister is provided by 25 support disks, which are located by equal spaces over the full length of the basket as shown in Fig. 1. The concrete module has 4 inlet vents and 4 outlet vents located on the both side-walls of the concrete module. Natural circulation derives the cooling air flow through air vents and carries the heat to the environment.

A 1/2-reduced scale model of a designed HCSM was used for thermal test, in which 24 rod-typed electric heaters were installed to simulate the decay heat of SNF.

Thermocouples were used for the temperature measurement. The measuring points were 9 for basket surface temperature, 3 for canister surface temperature and 4 for concrete module in middle section of the HCSM test model. Air temperatures of inlet and outlet vents were measured at 2 different points per vent.

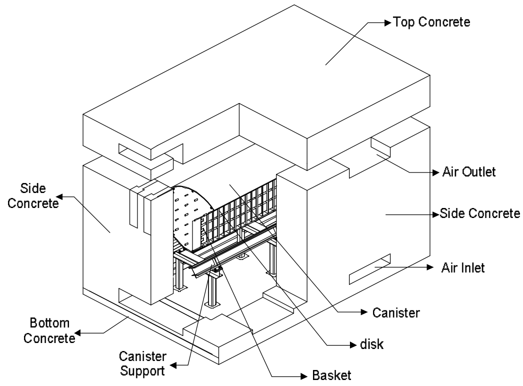

Fig. 1. Outline of the HCSM test model

The thermal test results under normal conditions as shown in Table I were chosen for validation.

Table I: Thermal Test Conditions

Conditions	Air Vent Condition	Ambient Temperature	Heater Capacity
Normal Condition (steady state)	100% opened	24℃	4.536kW

2.2 Analysis Model and Method

Fig. 2 illustrates computational grid of CFD model for HCSM test model. A half symmetric, 3-D. finite volume model for HCSM test model is developed using FLUENT version 6.2 [2].

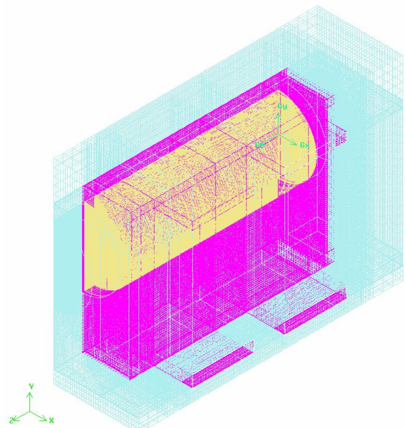

Fig. 2. Computational grid of HCSM test model

Flow regime inside and outside the canister is an important parameter that can affect the analysis. The assumption of fully turbulent flow inside and outside the canister will lead to a lower maximum temperature. On the other hand, assuming laminar flow inside and outside the canister would lead to a higher maximum temperature. Therefore a careful and correct consideration of the flow regime is required to avoid mis-prediction of the temperature field. In the present analysis, three different turbulence models (standard k-ε with standard wall functions, k-ε Renormalization-group (RNG) and transitional k-ω Shear Stress Transport (SST)) and laminar flow model inside and outside the canister, were used to model the air flow in flow regime. Generally, the standard k-ε model is suited to high Reynolds fully-developed flows, while k-ε RNG and transitional k-ω SST models are adequate to simulate the flow in low, transitional, and high Reynolds range [3].

3. Results

Temperature profiles resulting from CFD using four different models were compared to the test data. Fig. 3 and Fig. 4 show the temperature distributions for different turbulence models and laminar model inside the canister along with test data. As shown in Fig. 3 and Fig. 4, k-ε RNG and transitional k-ω SST models predicts the temperature distribution fairly well inside and outside the canister, while standard k-ε model over-estimates heat transfer at basket wall surfaces inside the canister than other turbulence models. It is also shown that when laminar model is used, basket wall temperatures are higher than other turbulence models and test data. The laminar flow regime over-predicts basket wall temperatures and is not appropriate to analyze the air flow inside and outside the canister. As a result, it can be concluded that flow condition inside and outside the canister would be flow in transitional and low Reynolds range. The results obtained from k-ε RNG and transitional k-ω SST models indicated that the problem was correctly modeled.

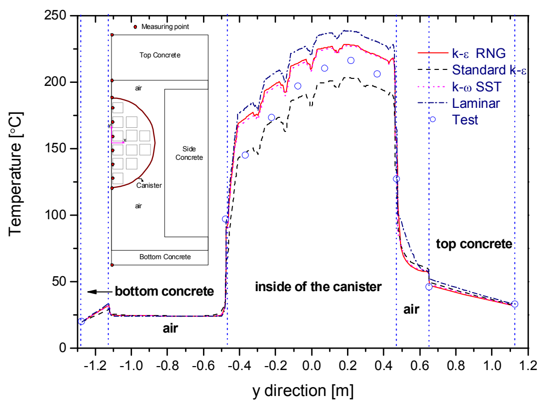

Fig. 3. Temperature distribution in middle section of the HCSM test model (y-direction)

As a result of analyses, the transitional k-ω SST and k-ε RNG models predicted the temperature distribution reasonably well in the basket region inside the canister as well as in the passage of cooling air, while the application of standard k-ε model with standard wall functions inside the canister led to too lower temperature distribution than test data.

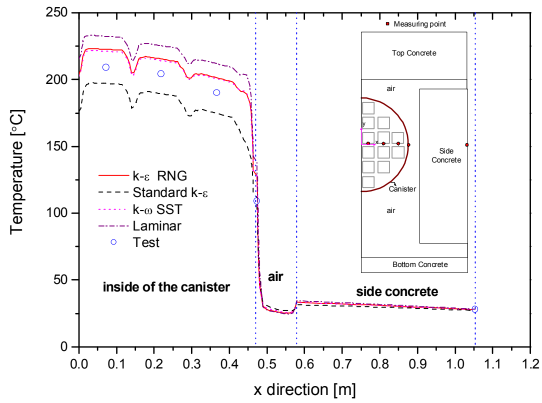

Fig. 4. Temperature distribution in middle section of the HCSM test model (x-direction)

4. Conclusions

Different turbulence models were investigated to validate the thermal model and method in the CFD simulation of the air flow and heat transfer inside and outside the canister.

Among various turbulence models, the validation demonstrated that the k-ε RNG and the transitional k-ω SST models predicted the measured temperatures closely and comparatively well. Finally, it can be concluded that the turbulence model of buoyancy driven flow is required to be applied correctly and uncertainties for turbulence modeling must be minimized.

REFERENCES

[1] Wataru, M., Takeda, H., Shirai, K. and Saegusa T., Thermal hydraulic analysis compared with tests of full-scale concrete casks, Nuclear Engineering and Design, 2008.

[2] FLUENT Computational Fluid Dynamics Software, Fluent, Inc., Centerra Resource Park, 10 Cavendish Court, Lebanon, NH 03766.

[3] Nicholas D. Francis, Jr., Michael T. Itamura, Stephen W. Webb, and Darryl L. James, CFD Calculation of Internal Natural Convection in the Annulus between Horizontal Concentric Cylinders, SAND REPORT, 2002.