Computational Fluid Dynamics and Heat Transfer Analysis of a Spent Fuel Dry Cask Storage System

Abstract

Excessive air and solid temperatures in spent fuel nuclear storage units can have catastrophic consequences. It is therefore critical that the cooling efficiency of such systems is accurately evaluated. Because of the complex physics involved in predicting local air patterns and heat exchanges inside the units, Computational Fluid Dynamics methods present significant advantages to perform a coupled flow and heat transfer analysis.

The current study demonstrates that a numerical solution can provide detailed temperature distributions on the canister, heat shields and concrete walls, as well as demonstrate good agreement with experimental measurements. Because CFD simulations are generally fast and inexpensive, they open new possibilities to evaluate the design of spent fuel storage units and improve the overall cooling efficiency of the system.

Introduction

Spent fuel nuclear storage systems must guarantee sufficient cooling to prevent excessive air and solid temperatures in the system and therefore avoid the possibility of catastrophic accidents. Because of the complex physical processes dictating flow conditions and heat transfer characteristics inside the unit, analysis requirements must include the ability to accurately predict free and forced convection, conductive and radiative heat transfer, and turbulence.

In the current study, special emphasis is placed on demonstrating that Computational Fluid Dynamics techniques can be used efficiently to determine local flow patterns in the air passageways and temperature distributions on the canister surface, heat shields and concrete walls. The numerical simulation is compared with experimental measurements reported by Pacific Northwest Laboratory for a 7 kW heating load in an existing NUHOMS storage system.

Computational Fluid Dynamics Model

The numerical model consists of four primary components: spent fuel canister, stainless steel heat shield, concrete enclosure, and air duct passageways. The geometry is decomposed into fluid and solid sub-regions with a total computational mesh size of approximately 120,000 cells (Figure 1).

For the purpose of this analysis, air is treated as an ideal gas and buoyancy effects are included using the Boussinesq approximation. Turbulent fluctuations in the flow are accounted for by using a standard k-e model with appropriate wall function treatment. The relevant physical properties for the different solid materials (density, thermal conductivity, and emissivity) are treated as constant over the expected range of temperatures. The thermal radiation energy exchanges between the solid surfaces are computed based on a view factor radiative heating model including internal obstructions.

The 7 kW heating load from the spent fuel is applied as a constant heat flux on the lateral canister surface and over the length of the fuel cavity. The axial heat flux through the end caps of the canister is assumed to be small and is therefore neglected. In this simplified model of the storage system, the heat exchanges within the canister, including shell conductivity, are not represented.

The CFD solution is executed using an automatic local time-stepping method until steady-state conditions are reached. The CFD software utilized for this analysis, STORM/CFD2000 (owned and developed by Adaptive Research), solves the full Navier-Stokes equations in a general curvilinear coordinate system. The code employs a structured grid technique and is based on a finite volume pressure-based approach applicable for all flow speeds [1]. The solution scheme is based on a strongly conservative formulation for the Navier-Stokes equations and uses a modified PISO procedure [2].

Computational Results

The CFD simulation provides significant insight on local flow patterns inside the storage unit, as well as detailed information on heat transfer characteristics for the entire system. Temperature distributions for the canister, heat shields, and concrete walls are shown in Figure 2. Air velocities at the symmetry plane and air temperatures for an axial cross-section are presented in Figure 3.

The numerical results shows that for a total heat load of 7 kW applied evenly over the fuel cavity length, the temperatures distributions at the symmetry plane are in good agreement with the experimental measurements (Figure 4) reported by Pacific Northwest Laboratory [3]. The slight discrepancies observed for the center region of the canister and concrete walls, as well as at the back end of the heat shield may be explained by some of the assumptions made in the numerical model (canister shell conductivity neglected, constant axial heat flux, no support rails). However, considering the uncertainty in the experimental data and lack of information on the exact positioning of the measurement devices, the numerical simulation performed well in predicting the overall temperature distributions for the storage system.

Conclusion

Overall, the numerical solution provides accurate details on local free and forced convection flow patterns, as well as conductive and radiative heat exchanges inside the storage unit. While improvements to the current model are needed to include heat transfer within the fuel cask, the results demonstrate that even with a simplified geometry, temperature distributions on the canister, heat shields and concrete walls are in good agreement with experimental measurements.

Acknowledgment

Adaptive Research would like to thank the Nuclear Engineering Division of Southern California Edison for their role in this study and the EPRI Organization for making available the experimental data used as reference for temperature comparisons.

References

1. "STORM/CFD2000 - Theoretical Background", Adaptive Research, 1997
2. Issa, R.I., "Solution of the Implicitly Discretized Fluid Flow Equations by Operator-Splitting," Journal of Computational Physics, 62, 40-65, 1985.
3. Pacific Northwest Laboratory, "Nuhoms Modular Spent-Fuel Storage System: Performance Testing", EPRI NP-6941, 1990.

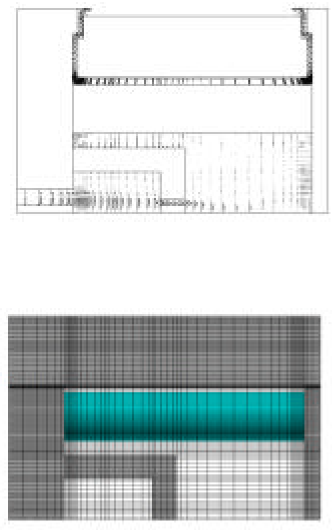

Figure 1. Airflow passages and computational mesh.

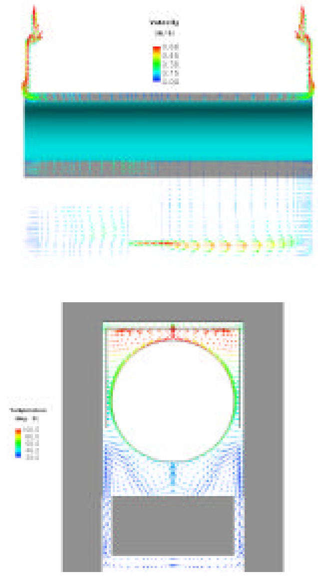

Figure 3. Velocity and temperature - symmetry plane.

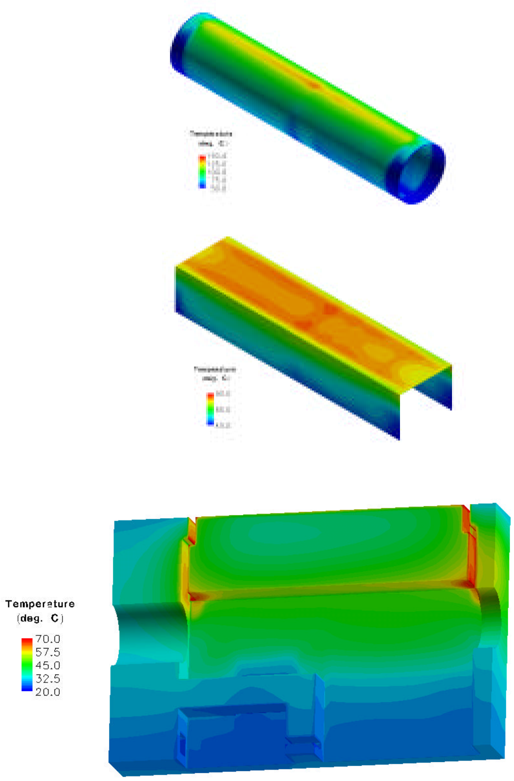

Figure 2. Canister/shields/concrete temperatures.

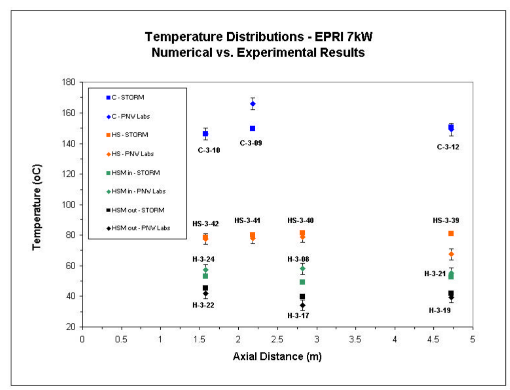

Figure 4. CFD vs. measured temperatures.

STORM/CFD2000 *is a registered trademark of Simunet Corporation.*