**Proceedings of the 19th International Symposium on the
Packaging and Transportation of Radioactive Materials
PATRAM 2019
August 4-9, 2019, New Orleans, LA, USA**

A Computational Fluid Dynamics Modeling Approach for the Design and Optimization of NUHOMS® MATRIX

Hui Liu, Jane He, and Venkata Venigalla

Orano TN Americas
Columbia, Maryland, USA

ABSTRACT

Due to the unavailability of nuclear fuel reprocessing and a permanent geologic repository in the United States, it has become increasingly essential for domestic nuclear reactors to have safe, secure, and efficient solutions for onsite long-term used nuclear fuel (UNF) storage. As existing UNF pools at reactor sites are approaching their capacity limits, dry storage facilities known as Independent Spent Fuel Storage Installations (ISFSI) has become an important option for utilities. Such systems are designed to passively reject decay heat during storage and transfer of UNF while maintaining thermal, structural and nuclear integrity.

Dry storage of UNF discharged from the reactor typically takes place 5 to 10 years after they are stored in the spent fuel pools. However, if a reactor is scheduled to be decommissioned, it is desirable to move the UNF into dry storage as soon as possible after the reactor is shut down. One of the key factors that limit the transfer of UNF to dry storage is the heat load. To provide flexibility in loading operations, NUHOMS® MATRIX (HSM-MX) systems, which is a two-tiered staggered, high-density horizontal storage module (HSM), is designed to reduce the footprint of the current EOS-HSM to allow for greater storage capability on an ISFSI pad than that currently available and cooling times required to move the UNF to dry storage. HSM-MX systems offer a wide range of thermal capabilities, including the industry-leading heat load of 50 kW per dry shielded canister.

This paper presents the design and optimization of HSM-MX based on a maximum heat load of 50 kW from 37 Pressurized Water Reactor (PWR) fuel assemblies. During the initial conceptual design process, the HSM-MX design was optimized using SolidWorks® Flow Simulation [1], an intuitive Computational Fluid Dynamics (CFD) tool embedded within SolidWorks® 3D, for quick evaluation of the thermal performance of various geometry designs. Then, the thermal performance of the optimized module is evaluated with a detailed CFD based modeling approach in ANSYS Fluent [2] code.

1.0 INTRODUCTION

The NUHOMS® MATRIX (HSM-MX) provides a staggered two-tiered self-contained modular structure for storage of used nuclear fuel (UNF) in an EOS-37PTH or EOS-89BTH Dry Shielded Canister (DSC). It is constructed from reinforced concrete and structural steel. Figure

1 shows a typical HSM-MX monolithic array at the Independent Spent Fuel Storage Installations (ISFSI).

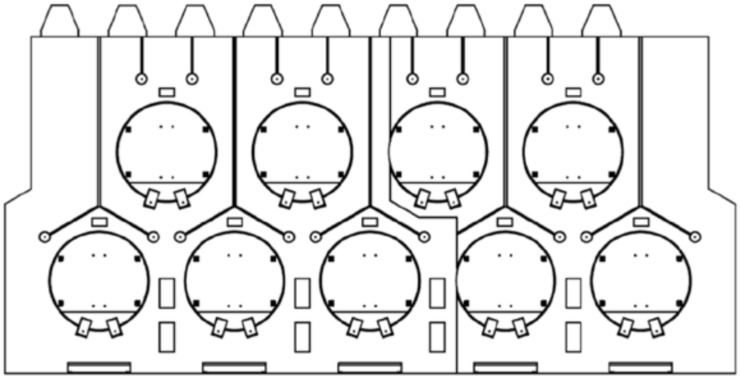


Figure 1: NUHOMS® MATRIX Monolithic Array

The HSM-MX System, as shown in Figure 2, includes two major components:

- A two-tiered staggered, reinforced concrete storage overpack designated as the HSM-MX that provides for the passive removal of the spent fuel's decay heat, environmental and seismic protection for the DSC, and radiological shielding from the DSC's contents, and
- A DSC that provides confinement, an inert environment, structural support, and criticality control for fuel assemblies.

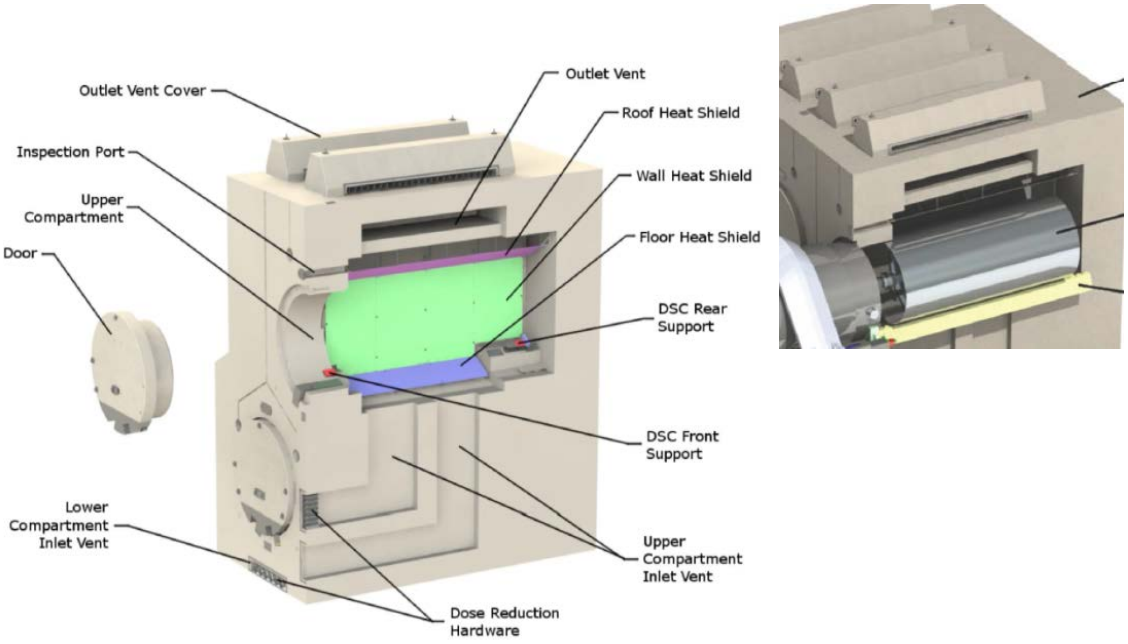


Figure 2: NUHOMS® MATRIX System Components and Structures

The HSM-MX provides a means of removing spent fuel decay heat by a combination of radiation, conduction, and convection. Ambient air enters the HSM-MX through ventilation inlet openings located on the lower tier of the HSM-MX, circulates around the DSC and the heat shields, then exits through the outlets of the HSM-MX.

Decay heat is rejected from the DSC to the HSM-MX air space by convection and then removed from the HSM-MX by natural circulation airflow. This passive convective ventilation system is driven by the pressure difference due to the stack effect (ΔP_s) provided by the height difference between the bottom of the DSC and the HSM-MX air outlet. This pressure difference is greater than the flow pressure drop (ΔP_f) at the design air inlet and outlet temperatures. Heat is also radiated from the DSC surface to the heat shields and HSM-MX walls and roof, where the natural convection airflow and conduction through the walls and roof aid in the removal of the decay heat. The outlet vent covers installed on the top of the HSM-MX are designed to mitigate the effect of sustained winds. The passive cooling system for the HSM-MX is designed to preserve fuel cladding integrity by maintaining the peak cladding temperatures of the used fuel assemblies below acceptable limits during long-term storage.

The staggered two-tiered nature of the HSM-MX design coupled with heat load of 50 kW (3.5 kW/assembly) presented a unique challenge for the designers. Due to the staggered nature of the HSM-MX design, each DSC compartment was designed to have independent heat dissipation mechanism while also being isolated from each other. Various design features are important to achieve the required thermal performance such as the heat shields and the vents, which were evaluated using an iterative mechanism in SolidWorks® Flow Simulation [1] and ANSYS Fluent [2].

2.0 CONCEPTUAL DESIGN

For more than 35 years, UNF storage has been limited to one-tiered horizontal or vertical storage. An idea of staggered storage concept was first received in TN's Idea Box, with the main intention to reduce the footprint size of the dry storage system by half, as shown in Figure 3 (A).

This idea was then converted to a preliminary design based on the honeycomb concept to have the most efficient use of space and materials and better shape for strength, as shown in Figure 3 (B).

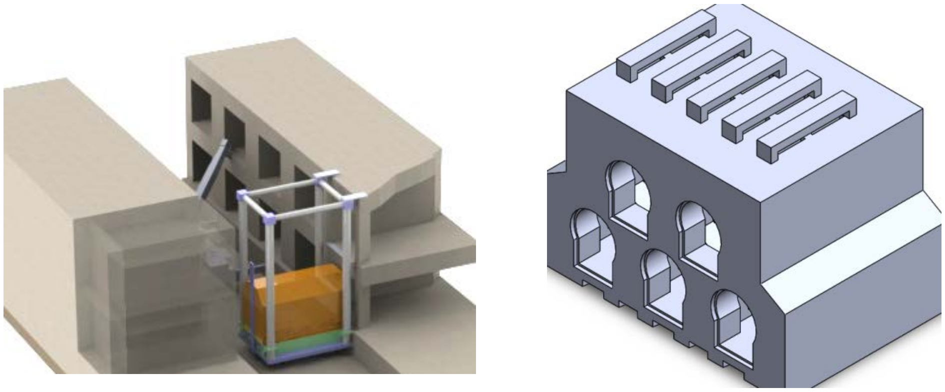
Based on this honeycomb concept, TN Americas started the development process for the innovative two-tiered spent fuel storage module designated as NUHOMS® MATRIX. Various optimizations have been completed during the design phases based on thermal, structural, and shielding performances. This paper only focuses on the optimizations for the thermal performance to achieve the highest heat load of 50 kW (3.5 kW/assembly) in industry. SolidWorks® Flow Simulation [1] is an intuitive CFD solution embedded within SolidWorks® 3D Computer-aided Design (CAD) tool, providing quicker and easier simulation while still delivering robust and highly accurate solutions. SolidWorks® Flow Simulation was chosen as the optimization tool because it can:

- Use native SolidWorks® 3D CAD data directly for CFD simulation,
- Handle complex geometry without needing to simplify the model,
- Generate automatic and effortless meshing for fluid and solid regions, and
- Modify CAD model and immediately analyze it without having to re-prepare the model.

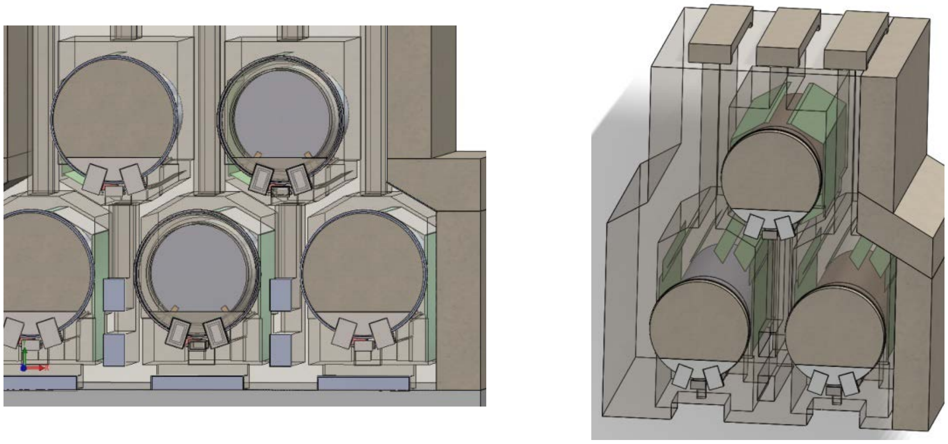
Thermal performance of a canistered UNF storage system, i.e., DSC stored in a concrete overpack is dependent on the thermal performance of both the DSC and the concrete overpack.

However, to isolate the thermal performance of the overpack, i.e., HSM-MX, the 37PTH DSC and its basket internals were not considered in the initial studies and a simplified model as shown in Figure 3 (C) was utilized. For the iterative evaluations, various parameters such as the temperatures in the upper and lower compartments and the concrete of the HSM-MX were considered along with the mass flow rates between at the inlet and outlet vents. One example of several design optimizations is the design of the inlet and outlet to achieve the best air flow distribution to reject decay heat from the DSC while also considering the impacts on shielding performance of the HSM-MX. Figure 3 (C) and (D) illustrate two designs during this optimization process.

Figure 4 presents the normalized temperatures through the iterative design process. As seen from the Figure 4, the temperatures within the upper and lower compartments were reduced by 15% compared to the initial design whereas the concrete temperatures were lowered by 35%.



(A) Initial Idea (B) Preliminary Design



(C) Intermediate Design (D) Design with 50 kW Heat Load

Figure 3: TN’s Initial Idea of Staggered Storage System for Used Nuclear Fuel

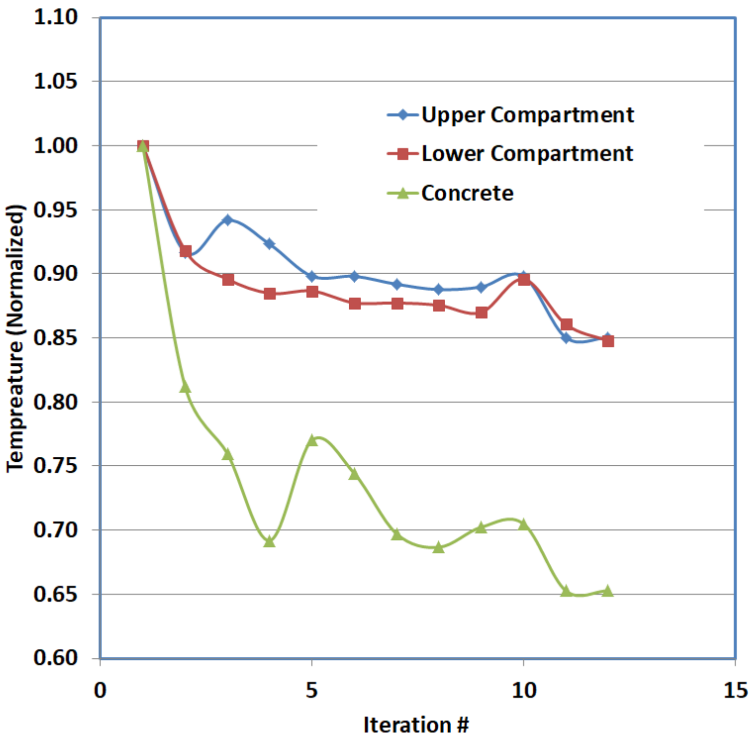


Figure 4: Normalized Temperatures within the Upper/Lower Compartments and Concrete

3.0 DETAILED CFD MODELING

While SolidWorks Flow Simulation is a powerful tool for the initial screening of various preliminary designs, ANSYS Fluent [2] and ANSYS ICEM CFD [3] are chosen as the detailed CFD modeling and meshing tools, respectively.

ANSYS Fluent [2] is a finite volume-based CFD software with broad physical modeling capabilities needed to model all modes of heat transfer and fluid flow in open-flow regions of a UNF storage system:

- Capability to model heat conduction, convection and radiation within the fluid and/or solid regions
- Inviscid, laminar, and turbulent flows
- Choices of different turbulence models
- Steady-state and transient flows
- Various types of boundary conditions including the customized subroutines
- Custom materials property database
- Scripting capability
- Parallel computing capability

With the finite volume method, the computational results crucially depend on the mesh used to discretize the computational domain. The resolution of the mesh should be fine enough to capture the important physical phenomena near the regions with higher gradients of temperature and flow velocity. Also, the mesh quality should be high to avoid introducing large numerical errors. ANSYS ICEM CFD [3] is chosen as the meshing tool because it is highly efficient and manipulative and allows the users to generate structured grids of high resolution, such as hexahedral elements. Hexahedral elements theoretically are the most efficient elements and are well suited for general CFD applications.

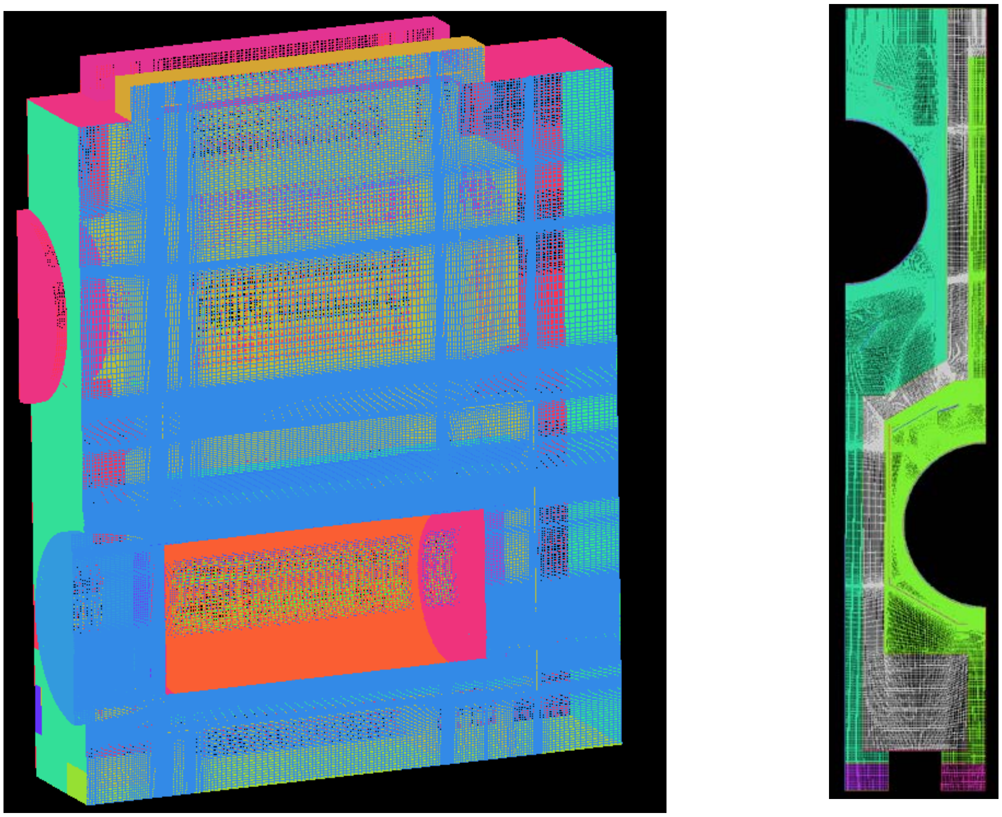
Meshing

Due to the complexity of the geometries, it is impractical to generate a completely conformal mesh for the whole model. Instead, some areas of the model are separately meshed in ANSYS ICEM CFD [3] using interfaces. Each set of mesh is created with conformal hexahedral elements by using the "blocking" method. The blocking approach breaks down a complicated geometry into small brick-shape domains, and structures the Cartesian mesh by the arrangement of the blocks. It provides a high degree of quality and control for hexahedral meshing.

To ensure the high quality of the mesh, the following guidelines on grids and grid design in NUREG-2152 [4] are considered in choosing the mesh parameters and techniques:

- Maintain the expansion ratio between two consecutive cells below 1.3 in the regions where high gradients of temperature and velocity are expected or material changes.
- Avoid highly skewed elements with angles less than 45 degrees or larger than 135 degrees, especially in critical regions.
- Minimize the number of nonmatching mesh interfaces, and avoid the mesh interfaces in critical regions.
- Keep the aspect ratio of most elements less than 20 except for those in the near wall regions.
- Use finer and high-quality mesh in critical regions with high temperature and velocity gradients or with significant changes in geometry, such as regions near the DSC outer surfaces, small helium gaps inside the basket assembly, etc.
- Ensure sufficient resolution in the near-wall regions adjacent to the wall to capture the large variations in the flow.

Figure 5 shows the isometric and cross-sectional views of the HSM-MX mesh through a transverse slice.



(A) Isometric View (B) Cross-sectional View

Figure 5: Isometric and Cross-sectional Views of the HSM-MX Mesh

CFD Modeling

To simulate the airflow within the cavity of the HSM-MX, the turbulence models based on low Reynolds adjustments as recommended in NUREG-2152 [4] is used. In addition to the convection within the HSM-MX cavity, the model also includes

- Heat conduction within the basket and HSM;
- Radiation heat transfer among the DSC shell, heat shields, support structure, and HSM;
- Solar insolation through the HSM front wall and roof; and
- Heat dissipation from the HSM and the vent outlet via convection and radiation to the ambient.

A pressure-based, double precision solver with segregated algorithm is used. As noted in Section 29.1 of ANSYS Fluent User's Guide [2], the pressure-based solver is applicable for a wide range of flow regimes from low speed incompressible flow to high-speed compressible flow. It also requires less memory and therefore is appropriate for the current evaluation. SIMPLEC scheme is used for pressure-velocity coupling. Second order upwind discretization method is used for the various variables.

The convergence of the steady-state CFD models in this analysis is closely monitored by several parameters such as the residuals, temperature variation, heat and mass balance.

4.0 RESULTS

Table 1 summarizes the maximum temperatures of fuel cladding and concrete of the HSM-MX loaded with the EOS-37PTH DSC at 50 kW heat load for all the bounding load cases.

- For the normal hot storage condition (Load Case 1), the maximum fuel cladding and concrete temperatures are 676°F and 229°F, within the temperature limits of 752°F and 300°F, respectively.
- For off-normal hot storage condition (Load Case 2), the maximum fuel cladding and concrete temperatures are 685°F and 245°F, within the temperature limits of 1058°F and 300°F, respectively.
- For accident blocked vents condition (Load Case 3), the maximum fuel cladding and concrete temperatures are 770°F and 355°F, within the temperature limits of 1058°F and 500°F, respectively.

Table 1: Maximum Fuel Cladding and Concrete Temperatures for Storage Conditions

Load Case	**Description**	**Max Fuel Cladding Temperature (°F)**			**Concrete Temperature (°F)**	
		Upper Fuel	**Lower Fuel**	**Limit(1)**	**Maximum**	**Limit(1)**
1	Bounding Normal Hot Storage Condition	637	676	752	229	300
2	Bounding Off-normal Hot Storage Condition	648	685	1058	245	
3	Bounding Accident Storage Condition	699	770		355	500

Notes:

(1) The temperature limits are from NUREG-1536 [5].

Figure 6 shows the streamlines for the airflow inside the HSM-MX loaded with the EOS-37PTH DSC under the normal hot storage condition. Cool air enters into the HSM-MX from the inlet, absorbs the heat from the EOS-37PTH DSC, and leaves the HSM-MX through the outlet with higher temperatures.

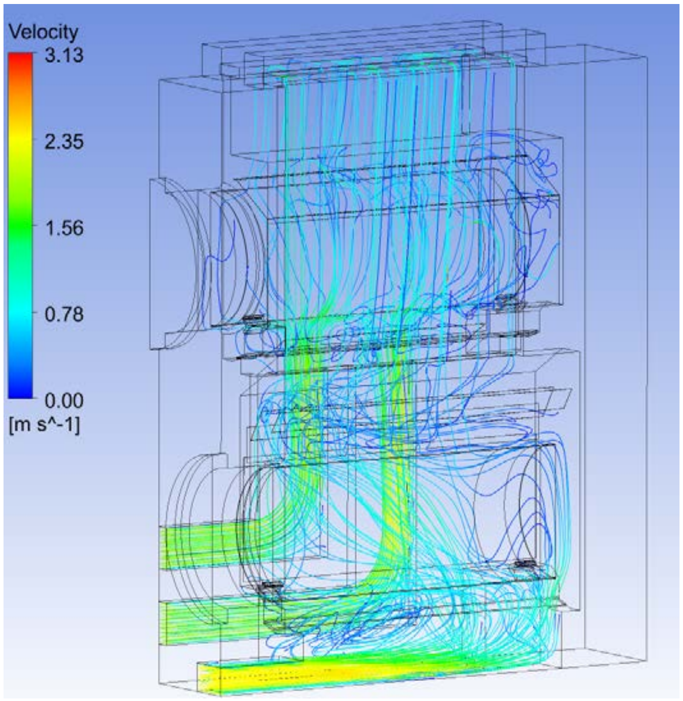


Figure 6: Streamlines of Airflow inside the HSM-MX Cavity

5.0 CONCLUSIONS

The two-tiered staggered NUHOMS® HSM-MX system is designed to reduce the footprint of ISFSI and to allow for a highest heat load of 50 kW per DSC in industry. HSM-MX is an innovative and robust dry storage solution for utilities including operating sites, shutdown sites and consolidated storage. The HSM-MX design was optimized for its thermal performance by using SolidWorks® Flow Simulation [1] for quick evaluation of various geometry designs during the initial design stage. Afterwards, the thermal performance of the optimized module is evaluated with a detailed CFD based modeling approach in ANSYS Fluent [2] code. The grid generation and modeling setups follow the CFD best practice guidelines for dry cask applications provided by NRC [4]. The results of the CFD analysis showed that all the thermal design criteria, including the maximum fuel cladding and concrete temperatures, are met for the NUHOMS® HSM-MX with 50 kW.

6.0 REFERENCES

1. SolidWorks 2011, Dassault Systèmes SolidWorks Corporation, June 2010.
2. ANSYS Fluent, ANSYS Fluent User's Guide, Version 17.1, ANSYS, Inc.
3. ANSYS ICEM CFD, Version 17.1, ANSYS, Inc.
4. USNRC, "Computational Fluid Dynamics Best Practice Guidelines for Dry Cask Applications," NUREG-2152, March 2013.
5. USNRC, Office of Nuclear Material Safety and Safeguards, "Standard Review Plan for Dry Cask Storage Systems - Final Report," NUREG-1536, Rev.1, July 2010.