Numerical Investigation to Examine Dust impacts on the Dry Cask passive Cooling under UAE harsh environmental conditions

**Alia Mohamed Al-Ghafri**  
Khalifa University of Science, Technology, and Research (KUSTAR)  
P.O. Box 127788, Abu Dhabi, UAE  
alia.alghafri@kustar.ac.ae

**Yacine Addad, Ho Joon Yoon**  
Khalifa University of Science, Technology, and Research (KUSTAR)  
P.O. Box 127788, Abu Dhabi, UAE  
Yacine.addad@kustar.ac.ae, hojoon.yoon@kustar.ac.ae

**ABSTRACT**

The present study aimed to numerically investigate the effects of the suspended dust in the air; on the passive cooling of the spent fuel dry storage system. First, a CFD model has been prepared for the ventilated storage cask VSC-17 using Star-CCM+ commercial code. Low-Reynolds k-ε turbulence model was employed to account for the turbulent air flow inside the cask. The present numerical predictions were compared with the experimental data of McKinnon et al. [2]. In contrast to numerical study reported in Zeigh et al. [11], an analytical step by step methodology was followed allowing to obtain the correct thermodynamic properties for the different solid material in the VSC-17 cask. As a result, the present predictions, were actually found in better agreement with the measured values in comparison with the previous numerical work.

Next, the validated model was used to examine the impact of the dust particles (nanoparticles) inclusion in the fluid air. The same turbulence model and boundary conditions, as applied in the base case, were used to run this case. The nanofluid properties were modeled using analytical formulas reported in the open literature. The dust concentrations that were tested in this study are 0.01% and 0.02%. Interestingly, the results obtained revealed that the presence of the dust, even with such relatively low concentrations, considerably enhances the heat transfer performance inside the dry cask.

1 **INTRODUCTION**

Safe management of Spent Nuclear Fuel (SNF) has become a major challenge for all countries employing the nuclear energy to produce electricity. The United Arab Emirates (UAE) will soon count amongst these countries, as the Emirates Nuclear Energy Corporation (ENEC) is on track to commission the first unit of Barakah Nuclear Power Plant (BNPP) in May, 2017 and the first discharge of SNF from this unit is expected to be in 2018 or 2019 depending on the refueling schedule [1].

The current ENEC’s plan is to start with the wet storage to keep the SNF in an onsite cooling pool where the borated water will be used to cool down the fuel and provides shielding from radiation. Based on this choice, the following key issues are to be considered:
• The storage racks design in BNPP pool is capable to safely hold up to 20 years of the SNF generation [1].
• With the continuous generation of SNF, the pool would get more packed and reach its full capacity.
• Recently, most of the nuclear organizations believe that these pools are potential targets for terrorist attacks as they are less protected than the reactor core.
• Since pumps are used to circulate the water between the pool and heat exchangers, the risk of losing cooling water due to emergency situations can’t be ignored especially after Fukushima Daiichi nuclear power plant accident and the SNF damage that took place in three of its units in March, 2011.

As a result, one is almost forced to accelerate transferring SNF to a safer storage facility. In contrast to pools, decay heat in the dry cask storage is removed by natural convection of the air travelling upward within the steel canister/cask gap as illustrated in Fig. 1 [2]. It is intuitive then to realize that the air movement inside the cask is driven by the buoyancy force. Accordingly, the colder the ambient air is, the larger this thermal difference becomes and the more efficient the corresponding heat removal is expected to be. For this reason, a feasibility study taking in account the harsh environmental conditions specific to UAE have to be carried out. These conditions are; the relatively high temperature, regular sand storms and a nearly permanent presence of dust in the atmosphere.

![Image of VSC-17 Cask components](image)

**Figure 1: VSC-17 Cask components**

2 LITERATURE REVIEW

A Number of high-precision experiments on heat transfer characteristics and velocity distribution for natural convection inside the dry cask were performed. Some of these experiments were used as reference data for the computational fluid dynamics (CFD) validation and verification (V&V).

Bang et al. [3], performed an experimental heat removal tests using a half scale dry storage cask with aim of estimating the heat transfer characteristics. The tests were conducted for normal, off normal and accident operating conditions. In the normal condition, the passive heat removal system is designed well and works efficiently since heat was dissipated to an ambient atmosphere through air natural convection at rate of 83%.

In 1992, the Pacific Northwest Laboratory and Idaho National Engineering Laboratory (INL) joined forces to conduct an experimental performance test on a Pacific Sierra Nuclear VSC-17 cask configured for pressurized water reactor (PWR) spent fuel [2]. The test consisted of loading the Multi – Assembly Sealed Basket (MSB) with 17 fuel assemblies containing...
consolidated spent fuel from Virginia Power's Surry reactors and Florida Power and Light's Turkey Point reactors. To examine the effect of the MSB Backfill Gas environment and Ventilation blockage, the performance test matrix included six different tests consisting of one cask orientation (vertical), four vent blockage conditions, and three backfill environment. To measure the temperature throughout the cask and evaluate its thermal performance during these tests, a total of 98 thermocouples (TCs) were used for instrumentation. Measurements for temperature were obtained at different radial points at elevation of 3.85m as well as measurements for axial temperature for the inner liner wall and the outer concrete surface were obtained.

Nishimura et al. [4], conducted a heat transfer and flow visualization experiment, using particle tracking velocimetry (PTV), in order to investigate the heat transfer characteristics and velocity distribution of natural convection flow inside a Dry Shielded Canister (DSC) containing 24 PWR spent fuel assemblies and filled with air or water at atmospheric pressure. The authors also conducted a numerical study in which a fifth-scale model was used to simulate the DSC. It was found that the average heat transfer coefficients were proportional to one fourth power of the Rayleigh number (∼Ra^1/4). The numerical runs showed that the computed flow patterns obtained from a two-dimension thermal hydraulic analysis provided a qualitative similarity to the experimental results within 8%, except at the top point of the center gap. However, the difference in the heat transfer coefficient was within 25% for air as the working fluid, while a much larger disagreement was obtained with the reference data when water was the working fluid.

In regards to numerical work on this topic, a number of studies have been reported in the literature. For example, Kim et al. [5] investigated heat transfer in a vertical concrete cask for PWR spent fuel dry storage. In addition to heat conduction and natural convection, the thermal radiation was also taken into account in their study. A full-scope CFD analysis of both; the prototype and model was performed to develop scaling laws between prototype and half-scale model of the dry storage. Yoo et al. [6] conducted a CFD analysis to investigate the thermal hydraulic phenomena in a Transnuclear TN-24 canister containing 24 PWR Westinghouse fuel assemblies by conducting a full-scope simulation using the Fluent commercial code. Heng et al. [7] conducted a numerical study to investigate natural convection heat transfer in horizontal spent-fuel storage cask with the aid of a CFD code. Both laminar and turbulent models were used. The numerical predicted results were compared with the experimental data reported by Nishimura et al. [4]. There was a good agreement between the experimental data and the numerical predictions using the laminar model (i.e. with no turbulence model considered in that run). Once a turbulence model was used, the numerical predictions returned Nusselt number (Nu) values higher than the experimental data.

Tseng et al. [8] investigated the thermal performance of a new tube-type dry-storage system (DSS) with 61 BWR (SNFs). In the study, both thermal radiation and conjugate heat transfer have been considered. A 3-D numerical CFD simulation using the k-ε turbulent model was carried out. It was shown that the maximum temperature is 333°C, and the minimum temperature margins are 81.5°C and 12.3°C for the fuel assembly and concrete structure, respectively. Hence, the results proved that the new DSS meets the NUREG-1536 requirements in terms of the fuel clad temperature and the material temperature limitation.

Das et al. [9] carried out a CFD simulation to investigate the flow and heat transfer in a ventilated concrete dry storage cask system-17 (VSC-17) using FLUENT software. Both normal operating conditions with opened inlets and outlets and off-normal condition with blocked vents were considered. The impact of the turbulence model on temperature distribution pattern was also studied. The turbulent models tested were the standard k-ω model, the
renormalization group k-ε model, the Shear Stress Transport (SST) k-ω model, and the realizable k-ε model. In all cases, there was an acceptable agreement between the computed results and the experimental data. It was noticed that the effect of the turbulence model used was negligible as the predictions with different k-ω and k-ε models returned similar temperature distribution.

Lee et al. [10], conducted a thermal analyses of a spent fuel storage cask numerically, using the finite volume commercial CFD code FLUENT, to predict the maximum concrete and fuel cladding temperatures. Normal and off normal conditions were considered in this analysis. Ambient temperatures of 27°C and 40°C are assumed for the normal and off-normal conditions respectively. A half blockage of the air inlet ducts is considered as an additional off-normal condition. Also a section in this study was dedicated to determining the optimum duct size and shape of the ventilation system. It was found from these analyses that under both normal and off normal conditions, the maximum temperatures of the fuel rod and concrete over pack were lower than the allowable values.

Zigh et al. [11] conducted a CFD simulation to evaluate the thermal performance of the VSC-17 under long term storage conditions. A 3-D CFD model for the VSC-17 was developed and validated using the experimental data reported in [2]. The flow in the channel was found to be in the transitional range between the laminar and turbulent regimes. Based on that observation, three turbulence models and a laminar model were applied. Among the chosen turbulence models, both transitional k-ω SST and low Reynolds k-ε predicted the measured temperatures fairly well. As both models are low-Reynolds, hence no use of wall-functions, finer mesh near the wall is required. Although the laminar model was not suitable to model the air flow, returning over-predicted temperature values within the PCT; it was found appropriate to model the helium flow inside the MSB as the helium gas trapped in the MSB is expected to have a very slow movement. It was claimed that the standard k-ε was a better choice than the laminar option but the model still over predicted the air flow heat transfer. However, in the report it was not mentioned if a different mesh was used for this high-Reynolds model with standard wall-functions but if not then the validity of these predictions is questionable.

As mentioned above, various experimental studies have been conducted to provide reference data ranging from simple temperature profiles to full velocity and temperature distribution using PTV techniques for instance. However, detailed information about the dry cask geometry is not reported in those studies making it difficult to build the dry cask related geometry and conduct a CFD validation. The only exception to this is the experiment conducted by McKinnon et al. [2] for which most of the geometry data is available in the open literature, hence the choice made to consider this case for the present study. Test #1, with helium gas in the MSB and fully open vents, corresponding to the normal dry cask operations, was selected to validate the present Star-CCM+ numerical predictions.

In regards to numerical studies, the review of previous work revealed that three-dimensional domain is necessary to correctly model the hydrodynamic and thermal behavior inside the dry cask. The influence of the turbulence model has been found negligible in some studies, while it was reported to have significant effects in others, hence this has to be further investigated. Relevant to UAE environmental conditions, it was shown that it is possible to overcome the effects of higher air temperature by improving the dry cask design [10]. On the other hand, none of the studies has considered the effect of dust on the passive cooling efficiency, thus this makes the current study the first of its kind.
3 NUMERICAL METHOD

3.1 Grid Generation and Numerical Approach

A 3-dimensional geometry was generated with a domain consisting of a quadrant of the air duct, Liner and the concrete parts as shown in Fig. 2. The heat load distribution reported in the experiment shows that the decay heat generation rate is almost the same in each quadrant. Based on that, symmetry was assumed at the edge of quadrants. A fine hexahedral mesh with 10 prism layers and with typical control volume size of $9\times10^{-2}$ m, resulting in a total number of 15,289,267 cells was employed. The maximum dimensionless normal distance, $y^+$, obtained with this mesh is 0.75 making it a valid grid to use with the low-Reynolds turbulence model of Lien et al. [12] employed in the current study. A representation of the hexahedral mesh is illustrated in Fig. 3. Since there is a temperature difference of more than 100°C, the thermodynamic properties of the air were defined as functions of temperature (see Tab. 1). An exception, is the specific heat capacity which was kept constant, $C_p = 1003.62 \text{kJ/kgK}$, as it was noticed to undergo a very small variation for the current temperature difference.

<table>
<thead>
<tr>
<th>Table 1: Air Thermal Properties</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density $(kg/m^3)$</td>
<td>$\rho = 3.1418 - 0.0084T + 7 \times 10^{-6}T^2$</td>
</tr>
<tr>
<td>Dynamic viscosity $(kg/m\cdot s)$</td>
<td>$\mu = 2 \times 10^{-5} + 6 \times 10^{-7}T - 3 \times 10^{-10}T^2$</td>
</tr>
<tr>
<td>Thermal conductivity $(kW/m\cdot K)$</td>
<td>$k = 10^{-6} + 10^{-6}T - 3 \times 10^{-10}T^2$</td>
</tr>
</tbody>
</table>

To start with, thermal properties for the VSC- 17 solid materials were defined the same as reported in the previous numerical study [11]. Also the same boundary conditions as detailed in [11] were prescribed in the present runs. The only exception is the MSB surface for which measured MSB outer wall temperature was applied as thermal boundary condition instead. This is due to the fact that the fuel part was excluded from the actual computational domain. Also in similar manner to the previous computational study, radiation between the inner cask walls was considered, using the discrete ordinate (DO) model, while solar insolation was not included since the measurements were performed indoor. As the simulation is run in steady state mode, the convergence criterion was satisfied once the residuals fall under the $10^{-4}$ limit. The second criterion used to assess convergence, are temperature variation probes placed at different locations in the air duct. These probes were used to monitor the temperature values to make sure that the steady state regime is reached.

3.1 Validation Test Case

Radial measured temperature profiles from the MSB outer surface to the concrete outer surface at elevation of 3.85m as well as axial measured temperature profiles for the inner Liner wall and the outer concrete surface, reported in [2], were used for the actual CFD validation section.

The initial predicted radial temperature profile is plotted in Fig. 4-a (case1). The parameters used in this case are the same as reported in the previous numerical study [11]. At first sight, the predicted temperature seems to be in a reasonable agreement with the experimental data. However, a close look in the predicted data shows that the temperature variation within the liner solid is somewhat over predicted. That is to say that numerical profile shows most of the heat to be conducted within the solid, while the experimental data gives a completely different picture. The explanation to this can be attributed to the fact that in the previous numerical study, a standard value for the steel conductivity was used.
Furthermore, a careful comparison between the experimental report and the previous numerical study, [11], revealed that also a lower concrete conductivity was used in the numerical study, while a much higher value was reported in the experiment. Due to these disagreements between the experimental report and the previous numerical work, it has been decided that before embarking in assessment of different turbulence models predictions and capabilities; first the correct solids properties should be prescribed.

Firstly, the correct value for the concrete conductivity is used. Then based on energy balance analysis using the experimental data; the steel conductivity was obtained and applied. Also a somewhat higher convection heat transfer coefficient, compared to the one reported in [11], was applied on the cask external wall. Indeed, prescribing the newly obtained values are observed to return a better temperature variation within the liner (steel) solid and an over-predicted but correct slope for the temperature variation within the concrete (see results for case2 in Fig. 4-a). However, as it can be noticed, the model is still over-predicting the liner inner wall temperature and temperature distribution through the concrete at that elevation. To improve the predictions in these regions, a modification of two steps was done. First, oxidation on the MSB outer wall was accounted for, hence the emissivity was reduced from 0.6 to 0.57. Using this somewhat lower emissivity values returns a much better prediction for the Liner inner wall temperature as shown in Fig. 4-b (case3). Finally applying a correct local heat transfer coefficient on the side of the VCC, which was obtained at that elevation using the energy balance calculations, returns now a much better predictions as illustrated in Fig. 4-b (case4). Tab. 2 summarizes the modifications applied in each case.

**Table 2:** The modifications applied in the four cases

<table>
<thead>
<tr>
<th>Thermal Property</th>
<th>Case1</th>
<th>Case2</th>
<th>Case3</th>
<th>Case4</th>
</tr>
</thead>
<tbody>
<tr>
<td>$K_{\text{concrete}}$ (W/m$^\circ$C)</td>
<td>1.47</td>
<td>1.96</td>
<td>1.96</td>
<td>1.96</td>
</tr>
<tr>
<td>$K_{\text{liner}}$ (W/m$^\circ$C)</td>
<td>41.5</td>
<td>1.33</td>
<td>1.33</td>
<td>1.33</td>
</tr>
<tr>
<td>$h$ (W/m$^2$-K)</td>
<td>6.95</td>
<td>6.95</td>
<td>6.95</td>
<td>9.2</td>
</tr>
<tr>
<td>$\varepsilon_{\text{MSB}}$</td>
<td>0.6</td>
<td>0.6</td>
<td>0.57</td>
<td>0.57</td>
</tr>
</tbody>
</table>

Figure 2: Schematic of the Computational Domain.  
Figure 3: Zoom in of the hexahedral mesh.
Fig. 5 shows that a reasonably good prediction is obtained for; the axial temperature profiles for the Liner inner wall, the concrete outer surface and MSB outer surface in comparison with the experimentally measured data.

![Radial Temperature Profile at z =3.85m](image1)

![Radial Temperature Profile at z =3.85m](image2)

(a) (b)

Figure 4: Radial Temperature variation across the air gap, liner, and concrete at z =3.85m.

![Axial Temperature Profiles](image3)

Figure 5: Concrete, Liner and MSB Axial Temperature Profiles

### 3.2 Numerical Runs with Dust Effects

Next, the validated CFD model generated for the VSC-17 was used to investigate the dust impact on the cooling efficiency. In these runs, the same physics models and boundary conditions as applied in case 4, are used. The nanofluid thermal properties were implemented assuming a single-phase fluid. Hence, the base fluid and the nanoparticles are assumed to have the same temperature and velocity field. Based on that, the solution of the continuity, momentum and energy equations can be obtained as if the fluid was a classical Newtonian fluid using effective properties of nanofluid. Accordingly, the effective properties of nanofluid are taken as a function of properties of both the constituents and their concentrations. These properties are determined using analytical models proposed in the open literature (see for
example models proposed in [13]). Two dust particles concentrations of 0.01% and 0.02% were tested and the heat transfer coefficient at the Liner inner wall and the MSB outer wall were compared for both cases.

In comparison of the base case (0% concentration) profiles and the dust cases (0.01% and 0.02%) are shown in Fig. 6. The figure clearly shows that the presence of the dust particles is considerably enhancing the heat transfer on the walls compared to the base fluid. Comparison between the averaged heat transfer coefficients on the walls for the three cases reveals that there is an enhancement by 12.8% and 12.5% on the MSB and liner walls respectively for the volume concentration of 0.01%. When a concentration of 0.02% is used, the heat transfer coefficient has increased by almost 20.4% on the liner inner wall and 22.8% on the MSB wall.

4 CONCLUSION

The objective of this study was to numerically investigate the dust impact on the passive cooling efficiency within the dry cask. A CFD model was prepared for VSC-17. The numerical predictions from the STAR-CCM+ code (version 9.4) were compared with the experimental data reported by McKinnon et al. [2]. A calibration approach was followed to get a good agreement with the measured values. Then, the validated model was employed to examine the effects of the dust inclusion. Two different dust concentrations have been tested, i.e. 0.01% and 0.02%. The results from these cases show that the addition of dust particles to the air flow results in a considerably enhanced heat transfer performance inside the VSC-17 cask. The main findings from this work are summarized as follows:

- In dry cask applications, radiation heat transfer is as important as convection heat transfer, hence both phenomena have to be included in the numerical simulations.
- A minimum of one quarter domain is required to correctly model the dry cask passive cooling.
- The present study has revealed that the existence of dust at low concentration within the air can actually enhance the dry cask passive cooling efficiency.

ACKNOWLEDGMENTS

All numerical runs have been performed in the AMD Opteron 1200 cores Khalifa University HPC cluster. First author is grateful for the MSc scholarship from the Emirates Nuclear Energy Corporation (ENEC).

REFERENCES


Figure 6: Heat transfer coefficient: (a) on the inner liner wall and (b) on the MSB outer wall