Simulation of Thermal Hydraulic Behavior of Dry Storage Cask Cooling During Convection Phenomena, Parametric Study

S. El-Naggar, R. Sakr*, M. Hejazy*, M. Elfawal and A. Gadalla

Nuclear fuel cycle Safety Department, Egyptian Nuclear and Radiological Regulatory Authority (ENRRA), Cairo, Egypt

*Faculty of Engineering - Benha University, Egypt

Received: 20/4/2016

Accepted: 25/6/2016

ABSTRACT

This paper is concerned with the investigations of the thermal hydraulic behavior during the cooling process of the spent nuclear fuel discharged from power reactors, which is stored in a dry storage cask containing 21 fuel assemblies during normal and accident conditions. An experimental test rig has been designed and constructed to simulate this case. The test rig is equipped with twenty one verticals electrical heaters made of steel to simulate the spent fuel. It is equipped with 38 thermocouples to measure surface and coolant temperatures. The test rig is used to investigate the effect of the decay heat variation on thermal-hydraulic behavior. The decay heat changed from 210 to 420, 630, 840 watt. Scenarios are made for an accident condition by blockage at the inlet and outlet of coolant. The ANSYS- CFX code has been used to simulate these effects. Comparisons between experimental and calculated results are performed.

Keywords: Dry storage/Spent nuclear fuel/ Thermal hydraulic/Blockage/ Decay heat

INTRODUCTION

The present study simulates the dry storage cask containing 21 spent nuclear fuel assemblies. It focuses on natural/and forced convection in the dry storage, cooling system under variation of some parameters such as heat generated per/rod, inlet coolant temperature and inlet coolant velocity. Spent fuel discharged from power reactors has two major problems first one it is highly radioactive and the second one it is very hot. It is cooled under water on the site of the reactor in the storage pool and for a period of time from 6 to 12 months after which the spent fuel may be sent to the reprocessing plant or maintained in the storage pool. As a result for “wait and see” the amount of spent fuel in interim storage facilities is increasing and the time needed for storage become longer and some problems for wet storage arises such as the problem related to the corrosion in clad of fuel, criticality and increase in storage cost. For this reason, the dry storage technique is considered to solve the problems of wet storage. After the spent fuels have decayed to an acceptable level of heat generation in storage pools (according to condition of fuel operation in a reactor), the spent fuel is stored in dry storage casks. Schematic drawing for the dry cask is shown in Figure (1) as given in (1). The spent fuel assemblies are stored in the core of the dry storage cask which is sectioned by neutron absorbing material to isolate each assembly. The perimeter of the cask, often has an annulus, which is filled with concrete. The perimeter of the cask is also lined with additional absorbent material if necessary to further decrease the level of radiation which escapes into the environment. For cooling the spent fuel, air inlets openings are located at the bottom of the cask whereby air cools the surface of the canister and go out from the top outlet opening. The present study focuses on the cooling of the cask and the parameters affecting this process.
Fig. (1): Schematic drawing for dry cask

**PREVIOUS WORK**

Various researchers investigated heat transfer by convection for dry storage cask. M. Nishimura et al. (1996)\(^{(2)}\) studied heat transfer characteristics for a horizontally placed fuel in dry storage cask containing 24 PWR fuel assemblies. They have used ZEPHYRUS code in the study. Their results show that Nusselt and Rayleigh numbers are approximately correlated by an equation of the form:

\[
\text{Nu} = 0.07 \text{Ra}^{0.27}
\]

at the range within \(10^6 < \text{Ra} < 10^7\), where Ra calculated according to equation:

\[
\text{Ra} = \frac{g \times \beta \times (T_s - T_i) \times R_C^3 \times P_f}{\nu^2}
\]

X. Heng et al (2002)\(^{(3)}\) investigated numerical heat transfer in a horizontally placed fuel in dry storage cask containing 24 PWR assemblies. They used PHOENICS-3.2 code. They used experimental data for comparison with code calculated results. The laminar and turbulent model are employed. The results show that laminar model agrees well, while the results of turbulent model are higher than experimental results.

C. Forsberg et. al (2003)\(^{(4)}\) experimentally and numerically studied thermal analysis for a cask with 21 spent nuclear fuel assemblies. They have studied the effect of liquid-cooled fins, which are added to the surface of the cask. The results showed the improvement in heat transfer coefficient.

V. Venigalla et. al (2007)\(^{(5)}\) used FLUENT code in two-dimensions to investigate the thermal performance of cask including four PWR fuel assemblies. They used helium and nitrogen as backfill gases. They used helium and nitrogen as backfill gases. The results showed that the allowable heat generation rate is 23% higher for helium than for nitrogen.

H. Murakami et. Al (2007)\(^{(6)}\) studied the effect of backfill gases numerically for dry storage Cask containing 21 fuel assemblies. They used vacuum, helium and nitrogen as a backfill gas. They used the FLUENT code at steady-state to calculate the peak cladding temperatures (PCT). The results showed that the vacuum environment is more challenging than the other gases.

K. Das et al (2010)\(^{(7)}\) made a numerical simulation of flow and heat transfer in a vertical storage cask containing 17 fuel assemblies, and compared the results with experimental data. Numerical simulations were
carried out in normal and off-normal conditions where both the inlet and outlet vents are blocked. The computed peak cladding temperature in all the simulation cases was slightly higher than the experimental data.

Y. Tseng et. Al (2011) (9) investigated numerically the thermal performance of a new tube-type of dry-storage system with 61 BWR spent nuclear fuels by utilizing the Computational Fluid Dynamics (CFD) code FLUENT. Results showed that the maximum temperature is 333 ºC for the fuel assembly. The results further demonstrate that the new tube-type of dry-storage system meets the thermal requirements in the NUREG-1536 guidelines (<400 ºC).

D. Gyu Lee et al (2013) (9) investigated numerically the thermal performance of full-sized model for the horizontally oriented metal cask containing 21 spent fuel assemblies. They used helium as a backfill gas under natural convection. The numerical predictions from simulations have been compared with the experimental data reported by Nishimura et al (2). The results showed that the average heat transfer coefficients are approximately correlated by \( \text{Nu} \propto \text{Ra}^{0.27} \) for laminar and turbulent model in the range of \( 3 \times 10^6 < \text{Ra} < 2 \times 10^7 \). Results agreed well with the experimental results for air as the working fluid. The heat transfer characteristics have shown that the Nusselt number is proportional to a 0.5 power of the Rayleigh number in the range of \( 1.5 \times 10^6 < \text{Ra} < 1.0 \times 10^7 \), while the Nusselt number is proportional to a 0.27 power of the Rayleigh number in the range of \( 1.8 \times 10^7 < \text{Ra} < 8.0 \times 10^7 \).

EXPERIMENTAL

The test rig, which has been designed and constructed to simulate the thermal hydraulic behaviour of a dry storage cask containing 21 spent nuclear fuel assemblies, is built in Nuclear FuelCycle Safety laboratory in the Nuclear and Radiological Regulatory Authority, Cairo. The test rig is designed with the following features and working capabilities:

- Variable diameter of cask to investigate the effect of variation of aspect ratio, \( B^* = \frac{D_{\text{cask}}}{D_{\text{canister}}} \).
- Variable Fan velocity to investigate the effect of inlet air velocity, \( V_{\text{in}} \).
- Variable power heaters at inlet to investigate the effect of Inlet coolant temperature, \( T_{\text{in}} \).
- Variable power heaters, which simulate fuel, to investigate the effect of heat flux of spent nuclear fuel, \( q^* \).
- Variable inlet and outlet opens dimension to investigate the effect of blockage at inlet and outlet.

The test rig consists of the following parts:

- Heaters (simulate fuel)
- Steel cylinder (simulates canister)
- Insulated cylinder (simulates cask)
- Steel sheet (to adjust heaters positions)
- Voltage regulator
- Temperature reading.
- Flow meter

Figure (2) shows a schematic drawing of the test rig, while figure (3) shows a photographic picture of the rig. Twenty-one electrical heaters are fixed in the vertical position to simulate the spent fuel. The heaters are arranged, as in actual cask, by using two steel sheets. The heat generated by heaters is transferred through the copper cladding to the air inside the steel cylinder and then to the steel cylinder. The coolant air at a temperature \( T_{\text{in}} \) enters the channel between the steel and isolated cylinders from four openings at the bottom, moving upward and leaves the channel at temperature \( T_{\text{out}} \). The cooled air removes the heat from steel cylinder. The test rig parts are housed in a box with a ventilation fan to control the inlet velocity of cooling air. Four heaters are situated at inlet of cooling air to control the inlet temperature. Two voltage regulators are used to control the power of electrical heaters and the heaters in the inlet air opening. Thirty eight thermocouples are used for measuring the temperature of heater surfaces, air channel, inlet and outlet coolant air. The recorded results from the experimental test rig include temperatures at the surfaces of heaters, and the steel cylinder. Temperatures in the flow channel, inlet air and outlet air have been also recorded. Heat transfer coefficient, velocity of the coolant and dimensionless numbers are calculated using values from experimental results and using the following equations as in [2]:

1. \( \text{Nu} \propto \text{Ra}^{0.27} \)
2. \( \text{Nu} \propto \text{Re}^{0.8} \)
3. \( \text{Pr} \propto \text{Re}^{0.3} \)
Heat transfer coefficient, $h$
\[ h = \frac{q}{(T_i - T_f)} \]

Velocity of coolant, $V$
\[ V = \frac{q}{A \times \rho \times C_p \times (T_{out} - T_{in})} \]

Reynolds number, $Re$
\[ Re = \frac{V \times R_c}{\nu} \]

Grashof number, $Gr$
\[ Gr = \frac{g \times \beta \times (T_s - T_c) \times R_c^3}{\nu^2} \]

Rayleigh number, $Ra$
\[ Ra = \frac{g \times \beta \times (T_s - T_c) \times R_c^3 \times P_r}{\nu^2} \]

Nusselt number, $Nu$
\[ Nu = \frac{h \times R_c}{k} \]

A numerical simulation of flow and heat transfer is performed by CFX code, the obtained calculation values are compared with experimental results.

**NUMERICAL SIMULATION**

Computational Fluid Dynamics (CFD) is a computer-based tool for simulating the behavior of systems involving fluid flow, heat transfer, and other related physical processes. It works by solving the equations of fluid flow (in a special form) over the region of interest, with specified (known) conditions on the boundary of that region. The set of equations which describe the processes of momentum, heat and mass transfer are known as the Navier-Stokes equations. These partial differential equations were derived in the early nineteenth century and have no known general analytical solution, but can be discretized and solved numerically. CFD can be used to evaluate the behaviour of a component at the design stage, or it can be used to analyze the difficulties associated with an existing component. This can lead to design modifications.
which can be tested by changing the geometry of the CFD model and seeing the effect. ANSYS CFX is a general purpose Computational Fluid Dynamics (CFD) software. It is used to investigate the thermal performance of cask under active and passive systems with the varied boundaries conditions. In this study the effect of changing the velocity, the heat flux and the inlet coolant temperature will be illustrated.

**Conditions and Assumptions**
1. Three dimension active and passive systems.
2. Steady state.
3. Upward flow.
4. Uniform heat flux

**Minimization of Calculation**

The computational domain consisted of a 1/8 of the whole circular cross section of the storage cask due to symmetry. This step reduces time and memory needed for calculations. Figure (4) shows the domain of calculations.

![Fig (4): domain of calculations](image)

**Numerical Model**

Figure (5) illustrates the model elements. The model consists of twenty-one heaters, air inside the canister, canister, air flow and insulated cylinder. Figure (6) shows meshing of cask model. The computational grid consisted of 463590 cells and 531860 nodes. The heaters is assumed to be at constant heat flux. The outer wall of insulated cylinder is assumed to be adiabatic. The flow air part is have two open used to inlet and outlet air. The interfaces between surfaces of parts are assumed as thin layer material and conservative heat flux.

![Fig. (5): Model of calculation](image)  ![Fig. (6): Mesh of cask model](image)
RESULTS

Effect of Heaters Power Change

Figure (7) illustrates the effect of heaters power variation on the temperatures distribution in simulated parts. It’s clear from the figure that the temperature of model elements increases as the heater power increase. Figure (8) illustrates the local temperature distribution in the hot rod (hot rod is the centeral heater, which is the hottest one of heaters). The figure presents a comparison between experimental and theoretical results. It shows also the variation of temperature due to change of each heater power in the range from 10 to 40 watt. It is clear that the increase in heater power leads to increase in surface temperature of the hot rod.

Fig. (7): Effect of heater power variation in the temperature distribution of model elements

Fig. (8): Effect of heater power variation in the temperature distribution of hot rod
Figure (9) illustrates a correlation between all calculated and experimental results, where it appears that the deviation between both lies in the range of 10 to 15%. In figure (10) the effect of heater power on the difference between outlet and inlet air temperature is illustrated.

![Fig. (9): A comparison between the experimental and code results for hot rod temperature](image)

![Fig. (10): Temperature difference between inlet air and outlet air](image)

Figure (11) shows the variation of the Rayleigh number with variation in heater power. The range of Rayleigh number is from 7.30E+06 to 2.11E+07. Figure (12) shows a comparison between experimental results, present correlation (by fitting experimental results) and Nishimura correlation for the relation between Nusselt number and Rayleigh number. It is clear from the chart that the present correlation predicts fairly well the experimental results. There is a difference between the present correlation and Nishimura correlation since the Nishimura correlation consider a horizontal cask while the present correlation consider the vertical cask. It is clear from the chart that Nusselt number for the vertical cask is greater than that for horizontal cask which mean that the heat transfer coefficient is improved by using vertical cask than a horizontal one. The present correlation is:

\[ Nu = 0.071 \times Ra^{0.362} \]
Fig. (11): Effect of heater power on Rayleigh number

Fig. (12): A comparison between experimental results, present correlation and Nishimura correlation for Nusselt number and Rayleigh number

Simulation of an Accident Scenario by Blockage at Inlet and Outlet of Coolant

Investigation of the effect of coolant blockage at inlet and outlet openings are performed experimentally using the test rig. The blockage ratio varies from zero, 25%, 50%, 75% and complete blockage at 100%. Also the calculated results for investigating the blockage of zero, 25%, 50% and 75% are obtained. The complete blockage cannot be studied using the calculation code due to the back flow effect. The visual investigation for temperatures of the cask at the varying percent of blockage are shown in figure (13). It is clear that by increasing the percent of blockage, the temperature inside the cask increases. Figure (14) shows a chart of the temperature of hot rod. It is a comparison between experimental results and the theoretical. It also shows the variation of temperature due to change of blockage percentage. The deviation between calculated and experimental results is in the range from -10% to 10 % as shown in figure (15).
Fig. (13) 3-D drawing for temperatures of model elements at the varying blockage percentage

Fig. (14): variation of hot rod temperature percentage of blockage
From the experimental results, the difference between the inlet and the outlet coolant temperature at the varying percent of blockage are shown in figure (16). It is clear from the chart that the differences increase with the increase of blockage percentage. The blockage at outlet cause more increase than at the inlet and the highest increase happen in complete blockage of the outlet.

**CONCLUSIONS**

- The results obtained by calculation present a good agreement with the experimental results. This agreement can be considered as a code validation for being used in another analysis.
- From the results discussed above, we conclude that the following correlation is much better in case of vertical casks than Nishimura correlation.

\[
Nu = 0.071 \times Ra^{0.362}
\]

- The case of blockage presents unfavorable issue as it leads to an increase in cask temperature, as it is noticed from the results. Outlet blockage leads to more temperature increase than inlet blockage. So it is recommended to follow the system by periodic maintenance.

**Recommendations for Future Work**

- Experimental and theoretical analysis will be used to investigate thermal hydraulic behavior of the cask during active operation of system at normal and accident conditions.
Experimental and theoretical analysis will be used to investigate thermal hydraulic behavior of the cask with different design parameters like aspect ratio.

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( h )</td>
<td>Heat transfer coefficient</td>
</tr>
<tr>
<td>( q'' )</td>
<td>Heat flux</td>
</tr>
<tr>
<td>( T_s )</td>
<td>Surface temperature</td>
</tr>
<tr>
<td>( T_c )</td>
<td>Coolant temperature</td>
</tr>
<tr>
<td>( V )</td>
<td>Coolant velocity</td>
</tr>
<tr>
<td>( q )</td>
<td>Heater power</td>
</tr>
<tr>
<td>( A )</td>
<td>Area</td>
</tr>
<tr>
<td>( C_p )</td>
<td>Specific heat of air</td>
</tr>
<tr>
<td>( T_{out} )</td>
<td>Outlet coolant temperature</td>
</tr>
<tr>
<td>( T_{in} )</td>
<td>Inlet coolant temperature</td>
</tr>
<tr>
<td>( Re )</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>( Rc )</td>
<td>Canister radius</td>
</tr>
<tr>
<td>( Gr )</td>
<td>Grashof number</td>
</tr>
<tr>
<td>( g )</td>
<td>Acceleration of gravity</td>
</tr>
<tr>
<td>( Ra )</td>
<td>Rayleigh number</td>
</tr>
<tr>
<td>( Pr )</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>( Nu )</td>
<td>Nusselt number</td>
</tr>
<tr>
<td>( K )</td>
<td>Thermal conductivity</td>
</tr>
<tr>
<td>Greek letters</td>
<td></td>
</tr>
<tr>
<td>( \rho )</td>
<td>Coolant density</td>
</tr>
<tr>
<td>( \nu )</td>
<td>Kinematic viscosity of coolant</td>
</tr>
<tr>
<td>( \beta )</td>
<td>Volumetric thermal expansion coefficient</td>
</tr>
</tbody>
</table>

REFERENCES