Thermal Analysis of Concrete Cask in Normal and Accident Conditions

A. A. Hassen, and K. Ahmed.
Nuclear and Radiological Regulatory Authority

Received: 3/1/2016 Accepted: 3/3/2016

ABSTRACT

Concrete cask used in the storage of spent nuclear fuel must be designed, erected and utilized safely according to appropriate standards and regulations. In this paper, a three dimensional finite element model simulating a quarter of concrete cask is established using the commercial Computational Fluid Dynamic (CFD) Code ANSYS to ensure that the system will satisfy the safety requirements. The analysis and the temperature distribution due to the air flow are analyzed in normal ambient temperature and at temperature of 800°C representing a fire accident condition for a duration of 30 minutes. Calculation results in the annulus region of air in normal condition are compared with test results for validation. Thermal analysis results in fire condition indicate small variation of temperature in the Stainless steel canister with a maximum variation relative to normal condition of 5.7°K and large variation in temperature distribution in the annulus region of air with a maximum variation from normal condition of 37°K. This variation will not exceed the safety limits.

Key Words: ANSYS / CFD / Finite element / Concrete cask

INTRODUCTION

Dry storage systems are preferred for a regional solution since they are easy to install and operate, open for all kinds of fuel, causing less secondary waste, simple for accident prevention and have the advantages of transportability, economical efficiency, safety, and public acceptance (1,2,3).

Concrete casks are used in USA, Japan and many other countries. Computational Fluid Dynamic (CFD) Codes have been recently used for design and study of different scenarios in normal and accident conditions for such cases (4).

A. Hussain and H. Sait (5) used CFD modeling to evaluate the temperature distribution in radial and vertical directions of the spent fuel cask. They obtained the temperature distribution inside the fuel basket using ANSYS CFD analysis to prove that the proposed design of the spent fuel storage cask is safe enough to keep the temperatures of the spent fuel well within the limits.

A. Y. Walavalkar and D. G. Schowalter (6) performed numerical simulation using the commercial CFD software FLUENT for the VSC-17 spent fuel rod dry storage system. Turbulence flow equations and energy equations with thermal radiation were solved for a 90-degree section of the VSC-17 system. Results of the simulation were compared with experimental data available. Comparisons of temperature profiles from simulation at various axial as well as radial locations with the experimental values were presented. Simulation results were seen to predict the temperature values observed experimentally with high accuracy in most of the flow domain and conservative results elsewhere. The validation showed that CFD can be an effective tool in the nuclear waste management area.

K.M.Pandey and Amrit Sarkar (7) studied the effect of thermal gradient and pressure difference between gap and coolant on a BWR fuel element. The fuel is made of uranium oxide pellets (UO2) and surrounded by a Zircaloy cladding. The gap between pellet and cladding was initially filled with helium gas. From the analysis, the authors found that though the pellet and cladding expanding due to thermal effect, it was within safe zone and the stress developed also in the permissible limit.
Masumi Wataru et al \(^{(8)}\) performed a numerical calculation for the concrete cask, they divided the model into two parts, inside and outside the canister, that were combined at the surface. They compared calculated results with test results and found that the analysis method was valid for normal and accident conditions.

In this study, thermal analysis is performed for concrete cask spent fuel storage system to ensure that the cask and fuel material temperatures will remain within the allowable limits for normal and accident conditions. A three dimensional finite element model of a quarter of concrete cask is established using the commercial Code ANSYS CFX 14.5\(^{(9)}\). Conduction and convection are considered the dominant modes of heat transfer in the cask. Calculation of the temperature field under normal condition and the transient analysis in 800°C fire for 30 minutes are carried out.

**Geometry of Concrete Cask**

The concrete cask is used in thermal analysis. Fig. (1) Shows outline of two types of concrete cask. The reinforced concrete cask type (left side Fig.) is chosen for the analysis. Table (1) represents the specifications of the concrete cask and the canister. Cooling air flows into the cask body through four inlets and raises up along the canister surface and ends through four outlets as shown in Fig. (1).

**Fig. (1): Outline of the Concrete Casks (reinforced concrete cask - left side & concrete filled steel cask – right side)**

<table>
<thead>
<tr>
<th>Specification of the concrete cask</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Height (mm)</td>
<td>5787</td>
</tr>
<tr>
<td>Outside diameter (mm)</td>
<td>3940</td>
</tr>
<tr>
<td>Inside diameter (mm)</td>
<td>1850</td>
</tr>
<tr>
<td>Weight (without canister) (ton)</td>
<td>150</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Specification of the canister</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Height (mm)</td>
<td>4630</td>
</tr>
<tr>
<td>Outside diameter (mm)</td>
<td>1676</td>
</tr>
<tr>
<td>Weight (with spent fuel) (ton)</td>
<td>35</td>
</tr>
<tr>
<td>Body material</td>
<td>Super stainless steel</td>
</tr>
<tr>
<td>Basket material</td>
<td>Stainless steel</td>
</tr>
</tbody>
</table>
Finite Element Model

The canister with the fuel elements are considered a homogenous heat source. Considering a boundary condition of constant heat flux in the canister the temperature field will be analyzed. One quarter of the cask has been chosen since the geometry is axi-symmetric. Natural convection heat transfer was considered using k-ԑ turbulence model \(^{(9)}\). Fig. (2) shows the domain of analysis.

![Meshes of Analysis Domain](image)

**Fig. (2):** Meshes of Analysis Domain

RESULTS AND DISCUSSION

**Normal Condition**

The boundary conditions for the model have been analyzed using canister surface test data of M. Wataru et al \(^{(8)}\). The results of temperature distribution as calculated by ANSYS in canister and the whole cask are shown in Fig. (3) and Fig. (4) considering the following initial conditions:

- Fuel elements power = 22.6 (KW)
- Inlet air temperature = 300 (оК)
- Inlet flow rate = 1.086 (kg/s)

Fig. (4) represents a comparison between the present Finite Element results obtained by ANSYS and test data of M. Wataru et al \(^{(8)}\). The results indicate a maximum of 6% difference in the temperature along the center of the annular region.

Fig. (5) represents the comparison between the present Finite Element results obtained by ANSYS and test data of M. Wataru et al \(^{(8)}\). The results indicate a maximum of 6% difference in the temperature along the center of the annular region.
Considering that the internal decay heat is the main heat source in the cask and that the heat dissipates by convection on the cask surface, the heat flux of dissipation can be written as

\[ q^{\star\star} = h_c (T - T_{\infty}) \]  

(1)

where, \( q^{\star\star} \) is the heat flux of dissipation.

\( h_c \) is the convection heat transfer coefficient.

\( T \) and \( T_{\infty} \) are the average temperature on the canister surface and the environment temperature respectively.

The convection coefficient can be calculated by the formula \(^{(10)}\),

\[ h_c = 1.3098 (T - T_{\infty})^{0.33} \]  

(2)

thus,

\[ q^{\star\star} = h_c (T - T_{\infty})^{1.33} \]  

(3)

The Fuel elements power is 22.6 KW, thus \( q^{\star\star} \) is 835.8 W/m\(^2\) then \( T \) is 439.5 °K

The average temperature on the canister surface obtained by calculation is 435 °K which is close to the theoretical value. The difference may be attributed to the geometry simplification.

**Accident Condition**

Transient analysis is carried out for 30 minutes under fire exposure at 800 °C. This is considered an accident condition.

To ensure that the canister temperature satisfy safety limitation, transient calculation is obtained for the fire condition. The temperature variation with time is obtained through four points at the hottest region in the canister. Figure (6) represents temperature variation with time at different points at the canister centerline. Points 1, 2, 3, 4, and 5 are at cask heights of 1, 2, 3, 4, and 5 m respectively. The maximum temperature variation is 5.7°K as shown in figure which is too slight to affect the thermal analysis.
The calculation results in the annulus region of air at fire condition indicates maximum temperature of 357°K for air compared to 320 °K for normal condition as shown in Figure (7).
CONCLUSION

For the thermal evaluation of concrete cask, the calculation method is developed using ANSYS 14.5 calculation code. The heat conduction analysis is obtained in canister region and thermal hydraulic analysis is obtained in the fluid region. The calculation results are compared with the test results for validation. Thermal analysis is performed for accident condition. Transient calculations in fire condition represent a small variation in the canister temperature distribution.

REFERENCES

(2) Donghak Kook, Jongwon Choi, Juseong Kim, and Yongsoo Kim, "Review of Spent Fuel Integrity Evaluation for Dry Storage” Nuclear Engineering and technology;45(1);(2013).
(3) IAEA-TECDOC-1100,,” Survey of Wet and Dry Storage”; July (1999).
(9) ANSYS workbench user manual, version 11, Providence RI, USA; (2007).