Arab J. Nucl. Sci. Appl., Vol. 52, 1, 145-153 (2019)

Arab Journal of Nuclear Sciences and Applications

Arab Journal of Nuclear Sciences and Applications

ISSN 1110-0451 *Web site: ajnsa.journals.ekb.eg* (ESNSA)

Investigation of the Thermal-Hydraulic Behavior of Dry Spent Fuel Casks under Forced Convection

S. Helmy, A. Hamza and S. Elnaggar
Nuclear and Radiological Regulatory Authority (NRRA), Cairo, Egypt

Received 25th Feb. 2018
Accepted 27th May 2018

The thermal-hydraulic behavior of the spent fuel in dry storage casks under forced convection mode is experimentally and numerically investigated. For this purpose, a test rig is designed and constructed to simulate the cooling loop cask. This test rig contains 21 spent fuel discharged from a pressurized water reactor (PWR). A numerical simulation is performed by ANSYS-CFX fluid dynamic code. The effect of decay heat generation and inlet air velocity are investigated. The results show that the increase in the inlet air velocity improves the coolability of the fuel, while the increase in decay heat leads to a decrease in the coolability of the fuel. Within the range ($1.1 < V < 2.8$ m/s) for inlet air velocity and heaters power ($630 < Q < 1260$ watt), a new empirical correlation has been obtained for Nusselt number, Nu as a function in Reynolds number, Re. The comparisons between experimental and numerical results show a good agreement.

Keywords: Spent fuel / Dry storage cask /Thermal hydraulic/ Forced circulation

Introduction

The spent fuel associated with the utilization of the nuclear reactors in electricity production is a major important problem facing the future of nuclear power. Long time storage of the spent fuel poses some problems including criticality and corrosion of spent fuel construction materials [1]. Due to the limitation of storage capacity and high running cost in wet storage system, there was an urgent need for adopting new types of storage techniques. Dry Storage Casks are designed to cool and store spent nuclear fuel assemblies for up to 100 years, after they have decayed to an acceptable level of heat generation in spent fuel storage pools. Figure (1) shows the outline of two types of dry concrete casks [8]. The perimeter of the cask often has an annulus which is filled with concrete. The perimeter of the cask is lined by a neutron absorbent material to decrease the level of radiation to environment.

Previous work

In the previous work, the majority of the researches were concerned with the natural convection heat transfer mode. The present work shows more interest in forced convection. Bang K. et al, [1], have performed experimentally the thermal analysis of dry storage and the results showed that in normal conditions, the convective heat transfer rate to the ambient reached 83%, and in abnormal conditions the influence of a half blockage of the inlet on the temperature appeared to be acceptable and, in the accident, conditions the temperature rise for twelve hours after the accident but after thirty six hours the rise is steady. Venkata et al. [2], have investigated numerically the thermal analysis of dry transport cask containing four spent fuel assemblies. The FLUENT software was used in the analysis of thermal performance in two dimensions. They used helium and nitrogen as a backfill gas.

Corresponding author: salwahelmy31@yahoo.com

DOI: 10.21608/ajnsa.2018.2995.1064

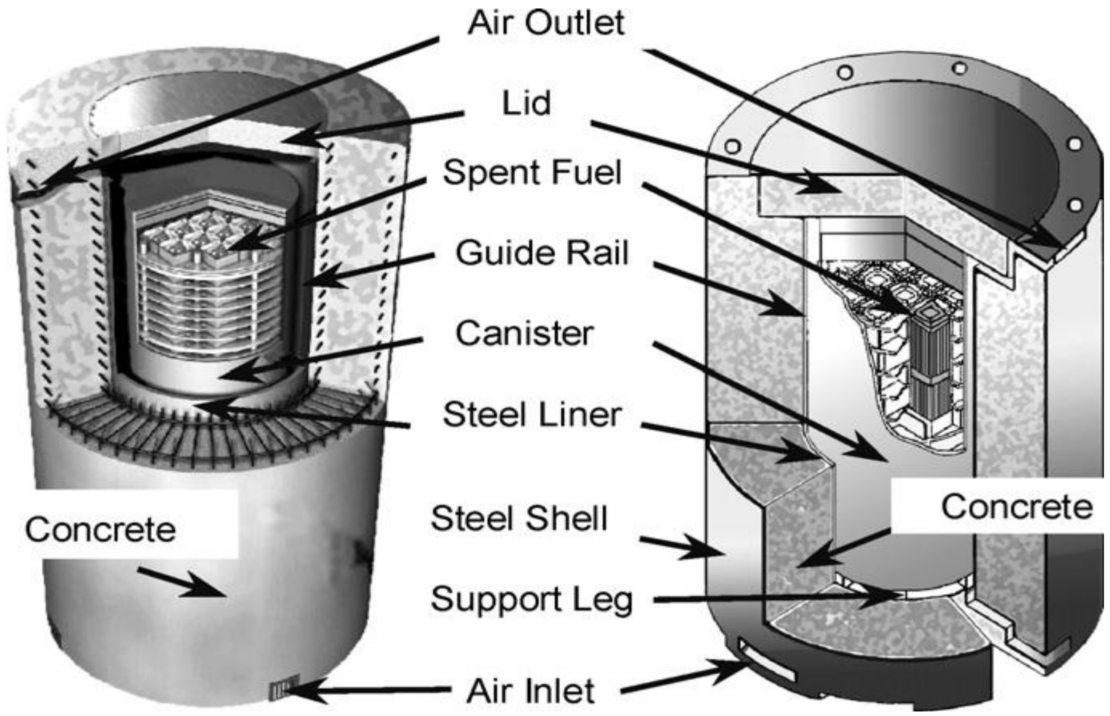


Figure (1): Outline of the concrete casks

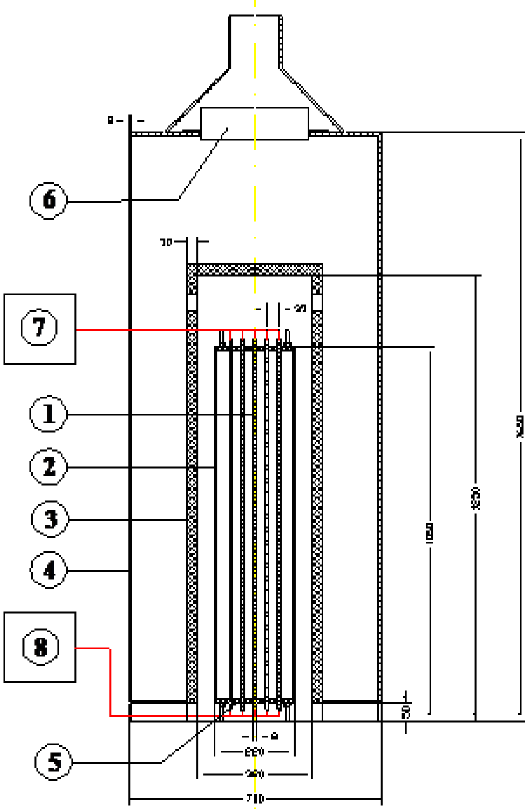
The results were compared with simulations, in which the gas speed is set to zero; the allowable heat generation rate is 23% higher for helium than for nitrogen. The peak fuel clad temperature calculated within limit for the heat capacity. Li Jie et al [3], investigated numerically the thermal analysis of dry storage cask. The FLUENT software used at steady state to calculate the peak fuel clad temperature. Calculations were performed with different heat generation fuel assembly. Also, they used vacuum, helium and nitrogen as a backfill gas. The results showed that the vacuum environment is more challenging than the other gas environments in that the peak clad temperature limit is exceeded at a lower boundary temperature for a given decay heat load of the spent fuel assembly. Wataru M. et al [4], have investigated experimentally and numerically the thermal analysis for a dry storage cask containing twenty one spent fuel assemblies discharged from PWR at natural and forced convection mode. The experimental results were compared with the code results and a good agreement for normal and accident conditions of the storage was found. Hussain and Sait [5], investigated numerically the thermal analysis of dry storage cask containing fifty four spent fuel assemblies discharge from CANDU reactor. The FLUENT software was used in the analysis. The temperature distributions in radial and vertical directions of the spent fuel cask were investigated. The ambient temperature is taken as 40°C. The results showed that the peak clad temperature of the fuel bundle was found to be 153°C which is below the temperature limit of 160°C.

A transport cask designed to carry used fuel assemblies was proposed [8]. Simulations were investigated for a cask with a smooth external surface and shield thicknesses. Another simulation was performed for a cask with a corrugated surface and a neutron shield thickness that satisfies shielding constraints. Temperature profiles indicated that a three-assembly cask with a smooth external surface will meet fuel cladding temperature requirements but will cause outer surface temperatures to exceed the regulatory limit. A cask with a corrugated external surface will not exceed the limits for both the fuel cladding and outer surface temperatures. Unsteady thermal processes in storage containers with spent nuclear fuel were simulated [9]. The daily fluctuations of outer ambient temperatures were taken into account. The modeling approach, which was based on the solving conjugate and inverse heat transfer problems, was verified by comparison the measured and calculated temperatures in outer channels. The time delays for reaching maximal temperatures for each spent fuel assembly were calculated. Results of numerical investigations showed that the daily fluctuation of the outer temperatures does not have a large influence on the maximal temperatures of stored spent fuel.

The experimental test rig

The test rig [10] consists of a steel cylinder which contains twenty one electrical heaters in vertical position. The heaters are arranged in such manner that simulates the actual case of dry storage cask by using two steel sheets. The steel cylinder, with heaters, was put in an insulated cylinder to simulate the concrete cask. The insulated cylinder has four cooling air inlets close to its bottom; as well as four openings close to the top used as outlets for air. The test rig is housed in an acrylic box equipped with a variable speed fan to control the inlet air flow velocity. The rig is also equipped with two voltage regulators to control the power input to the heaters. Thirty eight thermocouples are used to measure the temperatures at heaters surfaces, steel cylinder, air flow channel, inlet air and outlet air channel. A flow meter is used to measure the inlet air velocity. Four acrylic nozzles are used for directing the air inlet to air flow channel. One opening is made in the top cover of box to fix the fan in order to control the inlet air velocity. The test rig is illustrated schematically in Figure (2).

It is clear from the figure that the heat generated by the heaters is transferred through the clad to the air inside the steel cylinder and then to the steel cylinder. The heat is removed from the surface of steel cylinder by the air flow as a coolant. The air enters at temperature, $T_{in,}$ to the channel between the steel cylinder and the insulated cylinder from four openings close to the bottom, moving upwards and leaves the channel from four openings close to the top at temperature, T_{out}. Temperatures in the flow channel, inlet air and outlet air have been monitored. The heat transfer coefficient, velocity of the coolant and dimensionless numbers are calculated using values from experimental results and using the following equations [6]:



1 Heaters
2 Steel cylinder
3 Insulated cylinder
4 Acrylic box
5 Circular steel sheet
6 Variable speed fan
7 Voltage regulator
8 Temperature recorder

Figure (2): A schematic diagram of the test rig

- Convection power, Q_{conv}

  $Q_{conv} = Q_t - Q_{rad}$

  Where:

  $Q_{rad} = \varepsilon\sigma A(\overline{\mathbf{T}_{\mathbf{s,o}}}^4 - \overline{\mathbf{T}_{\mathbf{a}}}^4)$

- Heat transfer coefficient, h

$$h = \frac{Q_{conv}}{A \times \Delta T}$$

- Nusselt number, Nu

$$Nu = \frac{h(D_{ins} - D_{s,o})}{\lambda}$$

- Modified Grashof number, Gr*

$$Gr* = \frac{g\beta q_{conv}(D_{ins} - D_{o,s})^4}{\lambda\upsilon^2}$$

- Modified Rayleigh number, Ra*

$$Ra* = Gr* \times \Pr$$

- Reynolds number, Re

$$\mathrm{Re} = \frac{\rho V(D_{ins} - D_{s,o})}{\mu}$$

Mathematical model

Problem description

The physical configuration which describes the problem is illustrated in Figure (3). The assumptions of calculation are an incompressible fluid, steady and neglecting viscous dissipation. The model consists from the following parts:

- Inside the steel cylinder, the heat generated by heaters is transferred by natural convection and radiation to the inner surface of steel cylinder.
- The heat is transferred by conduction through the steel cylinder until it reaches the outer surface of steel cylinder.
- The heat is transferred by forced convection and radiation to the air flow.
-

Governing equations

In the following analysis, the air flow is assumed to be in the steady state, laminar and incompressible with negligible axial conduction and viscous dissipation. The governing equations for mass, momentum and energy conservations can be expressed as follows [7]:

- **Conservations of mass – the continuity equation**

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

- **Conservations of momentum – the momentum equation**

x-direction

$$u\frac{\partial u}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial w}{\partial y} = -\frac{\partial p}{\partial x} + u\frac{\partial^2 u}{\partial x^2} + v\frac{\partial^2 u}{\partial x^2} + w\frac{\partial^2 u}{\partial x^2}$$

y-direction

$$u\frac{\partial u}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial w}{\partial y} = -\frac{\partial p}{\partial y} + u\frac{\partial^2 u}{\partial x^2} + v\frac{\partial^2 u}{\partial x^2} + w\frac{\partial^2 u}{\partial x^2}$$

z-direction

$$u\frac{\partial u}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial w}{\partial y} = -\frac{\partial p}{\partial z} + u\frac{\partial^2 u}{\partial x^2} + v\frac{\partial^2 u}{\partial x^2} + w\frac{\partial^2 u}{\partial x^2}$$

- **Conservations of energy - the energy equation**

$$u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z} = \alpha\left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right]$$

The relation between Cartesian and Cylindrical coordinates is represented as follows:

X = r x cosϑ, Y = r x sinϑ and Z = Z

The partial differential equations, boundary conditions and assumptions give a mathematical description of the problem.

Numerical method for solving the governing equations

The governing equation, the partial differential equations form cannot be solved analytically, therefore numerical methods are needed. Any numerical method involves two basic steps: first, the differential equations must be transformed to a set of algebraic equations by discrediting them and secondly, this set of algebraic equations is solved.

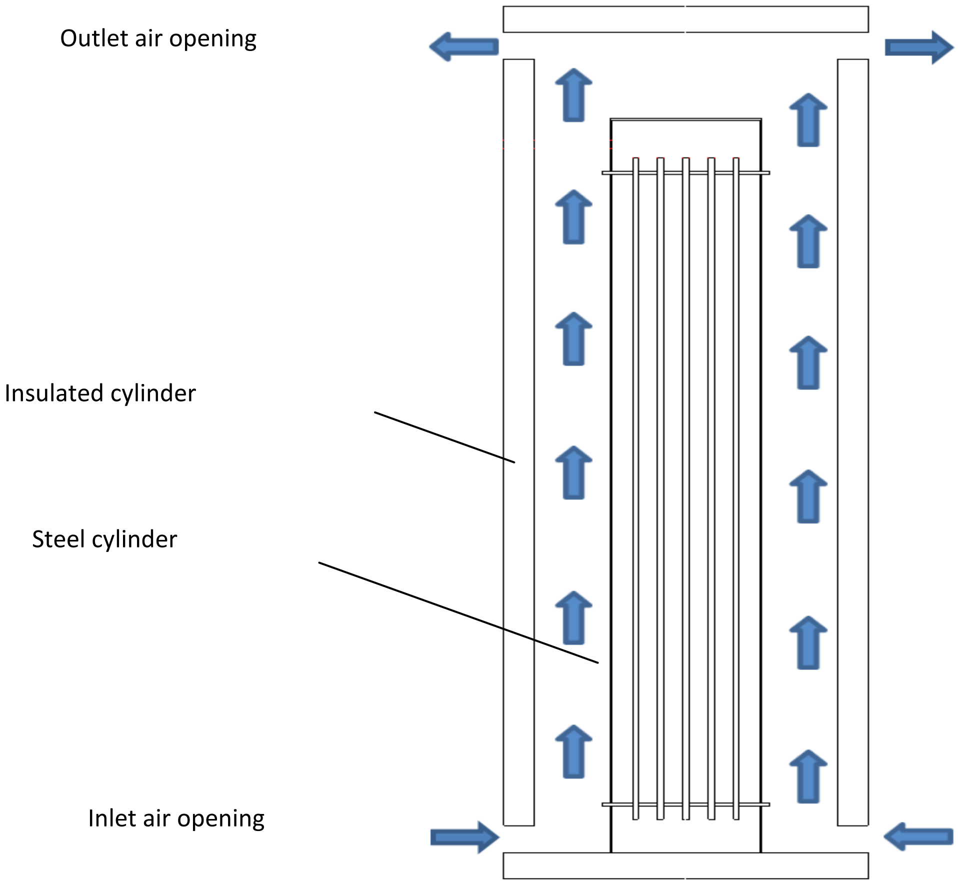


Figure (3): Schematic diagram of heat flow types in cask

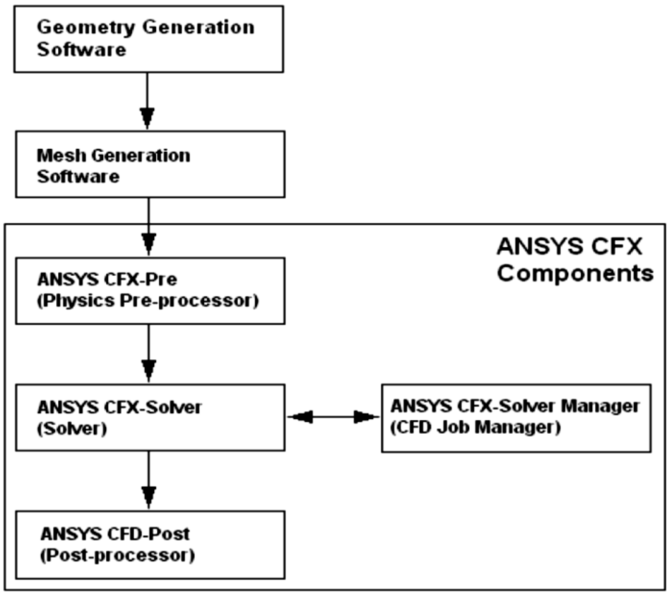


Figure (4): Flow chart to perform an ANSYS CFX analysis

Numerical simulation

ANSYS CFX is a general purpose Computational Fluid Dynamics (CFD) software which is used in theoretical analysis of the present studied. Figure (4) shows the flow chart for performing a ANSYS CFX. It works by solving the set of equations which describe the processes of mass, momentum and heat transfer in discretization form and solved it numerically. The process of performing a single simulation is split into four components:

1. Creating the Geometry/Mesh
2. Defining the Physics of the Model
3. Solving the Problem
4. Visualizing the Results in the Post-processor

Minimization of calculation

The pre step of using the code is minimizing the computational domain. It consisted of 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 (5) shows the domain of calculations.

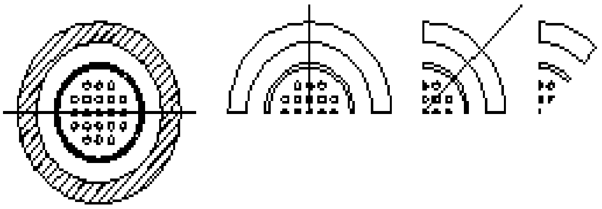


Figure (5): Domain of calculations

Result analysis And Discussion

The experimental and theoretical results are classified into two groups according to the model of heat transfer of air cooling outside the steel cylinder to natural or forced convection. Natural convection was discussed and evaluated before by Bullard et al. [8]. The Forced convection cooling system results are investigated in the present work.

The effect of heaters power, Q variation:

The total power is changed to have values of 630, 840, 1050 and 1260 watts. Considering the following parameters are constant; inlet air temperature (Tin ≈ 20 ^{o}C), inlet air velocity (V = ٢.٨ m/s) and aspect ratio (B* =10). The results of these changing effects are shown in the following:

a) The effect of input heaters power variation on the local and average central heater temperature are shown in Figs. (5 and 6). From these figure, it could be noticed that the increase in the input heaters surface power leads to an increase in the local central heater temperature. This increase could be due to the increase in the heaters power which leads to the increase in temperature difference between heaters surfaces and air inside the steel cylinder which leads to an increase in surface temperature of heaters. Also, the figure shows that the local temperature increases with the increase of height up to about 0.8 of heater height and then decreases above this height. This decrease above 0.8 of the heater height could be attributed to the increase in cooling air flow rate above the heaters. This increase in air flow rate fastens the heat removal. Good agreement between the experimental measurements and the theoretical data is shown in Figure (6), where the average deviation between the ANSYS-CFX code and the experimental results is about 8%. The deviation is due to neglecting some variables in the ANSYS-CFX code such as frictionless, heat losses, material properties, leakage of air from steel cylinder …etc.

b) The relation between the average Nusselt number for heater and modified Rayleigh number is demonstrated in Figure (7). It's clear from the figure that the average Nusselt number increases with the increase in the modified Rayleigh number for heater.

c) The relation between the average Nusselt number for steel cylinder and the Reynolds number is demonstrated in Figure (8). It's clear from the figure that the average Nusselt number increases with the increase of the modified Rayleigh number for the heater.

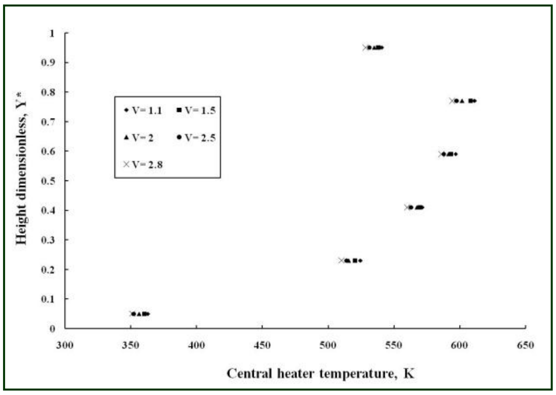


Figure (9): Local central heater surface temperature Vs height dimensionless, for different inlet air velocity

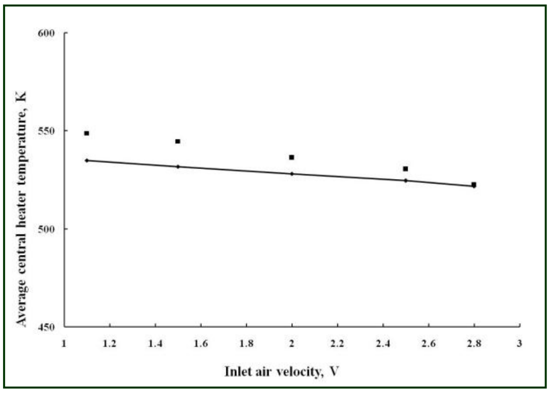


Figure (10): Comparison between Experimental and code results for average central heater surface temperature Vs inlet air velocity

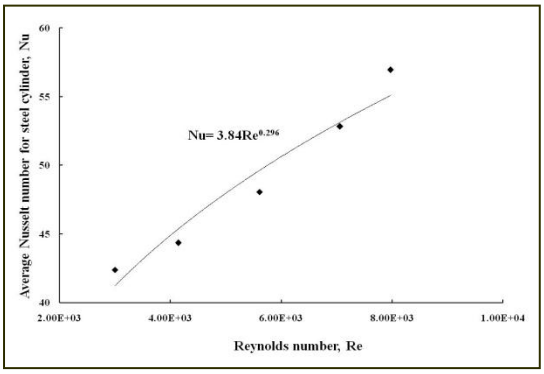


Figure (11): Average Nusselt number for heater Vs modified Rayleigh

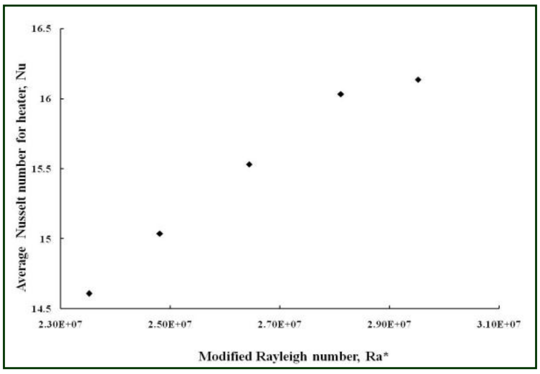


Figure (12): Average Nusselt number for steel cylinder Vs Reynolds number

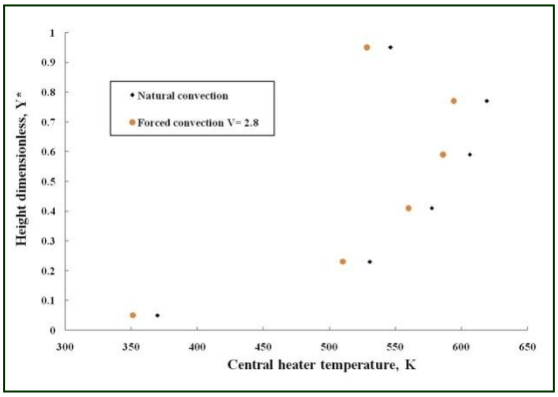
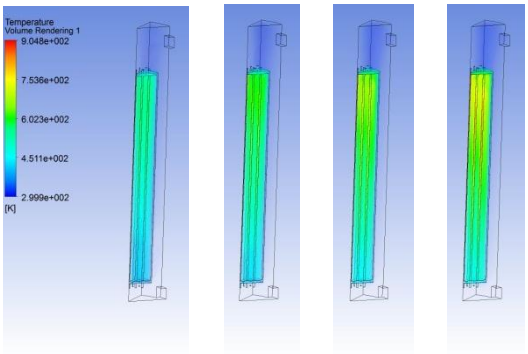
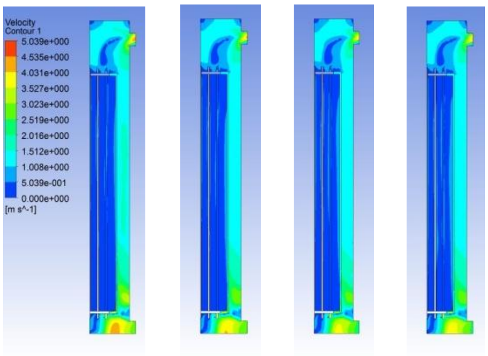


Figure (13): comparison between the experimental results for central heater surface temperature Vs height dimensionless, during Natural and Forced convection mode.



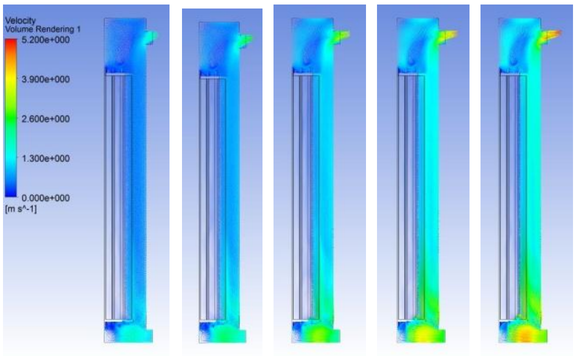
Q = 630 w 840 w 1050 w 1260 w

Figure (14): A schematic drawing for the effects input heater power variation on the temperature distribution of model elements



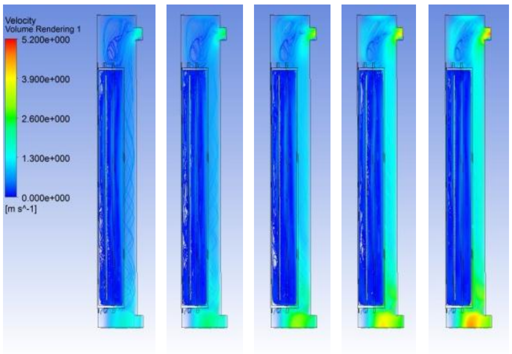
Q = 630 w 840 w 1050 w 1260 w

Figure (15): A schematic drawing for the effects input heater power variation on the contour in the symmetric plane for the velocity distribution.



V = 1.1 1.5 2 2.5 2.8

Figure (16): A schematic drawing for the effects of inlet air velocity variation on the velocity vectors of flow air at outlet of channel



V = 1.1 1.5 2 2.5 2.8

Figure (17): A schematic drawing for the effects of inlet air velocity variation on the streamlines velocity of flow air

Visual investigations for the results

The post-processor is the component used to analyze, visualize and present the results interactively. The visual investigations for temperatures of the cask at the varying input heaters power are shown in Figure (14). It is clear from the figure that increasing the input heaters power leads to increasing the temperature inside the cask. The visual investigations for velocities of air inside the cask are shown in Figure (15). It is clear from the figure that increasing the input heaters power leads to increasing the velocity inside the cask.

Visual investigations for velocities of air inside the cask at varying inlet air velocities are shown in Figure (16). It is clear from figure that increasing the inlet air velocity leads to an increase in the velocity inside the cask. Also, this effect is shown in Figure (17) which illustrates the streamline at the cask.

Conclusion

Simulation of the thermal hydraulic behavior of the spent fuel discharged from PWR, in dry storage cask by forced circulation mode is investigated experimentally and numerically by CFX code. The effect of heat generation and inlet coolant velocity on the local central intermediate and outer heaters temperatures, local air flow temperature are in good agreement with the corresponding theoretical computational results obtained using the ANSYS - CFX code.

Within the range ($1 < V < 3$ m/s), the following new empirical correlation has been obtained for Nusselt number for the steel cylinder, Nu as a function in Reynolds number, Re.

$$Nu = 3.84\, Re^{0.296}$$

References

1- K. Bang, J. Lee, K. Seo, C. Cho, S. Lee and J. Kim, "Heat Removal Test Using a Half Scale Storage Cask" Nuclear Engineering and Technology, Vol.39 No.2 April 2007.

2- V. Venkata, R. Venigalla and M. Greiner, "CFD Simulations of Natural Convection / Radiation Heat Transfer Within the Fuel Regions of a Truck Cask for Normal Transport" ASME Pressure Vessels and Piping Division Conference, San Antonio, Texas ,July 22-26, 2007.

3- Jie Li, H. Murakami, Y. Liu, P. Gomez, M. Gudipati, and M. Greiner, " Peak Cladding Temperature in a Spent Fuel Storage or Transportation Cask", The 15th International Symposium on the Packaging and Transportation of Radioactive Materials, Miami, Florida, USA,October 21-26, 2007.

4- M. Wataru , H. Takeda, K. Shirai and T. Saegusa, "Thermal Hydraulic Analysis Compared With Tests of Full-Scale Concrete Casks", Nuclear Engineering and Design 238, 1213–1219, 2008.

5- 5-A. Hussain and H. Sait, "CFD Modeling of The Dry Spent Fuel Storage of a Pressurized Heavy Water Reactor", Mathematical Theory and Modeling, Vol.2, No.10, 2012.

6- M. Nishimura, H. Shibazaki, S. Fujii and I. Maekawa, " Natural Convection Heat Transfer in the Horizontal Dry Storage System for the LWR Spent Fuel Assemblies", Journal of Nuclear Science and Technology Vol. 33, No. 11, p. 821-828, 1996.

7- S.N. Yu, J.K. Lee, B.S. Park, I.J. Cho and K.H. Kim "Hot Cell Renovation in the Spent Fuel Conditioning Process Facility at The Korea Atomic Energy Research Institute", Nuclear Engineering and Technology, 47, 776-790.

8- T. Bullard, , M. Greiner, M. Dennis, S. Bays and R. Weiner "Thermal analysis of proposed transport cask for three advanced burner reactor used fuel assemblies", Packaging, Transport, Storage & Security of Radioactive Material, Vol. 21, No 3, 2010.

9- S. Alyokhina and A. Kostikov, "Unsteady Heat Exchange at the Dry Spent Nuclear Fuel Storage",Nuclear Engineering and Technology, 49, 1457-1462, 2017.

10- S., A., El-Naggar ., “Simulation of the Thermal-Hydraulic Behaviour of Nuclear Fuel Storage",. Ph.D Thesis, Faculty of Engineering at Shoubra, Benha University, Cairo, Egypt 2017.