International Journal of Heat and Mass Transfer 128 (2019) 431–442

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Local heat transfer characteristics of natural circulation flow inside an 8 × 8 partial spent fuel assembly under dry storage

Doyoung Shin [a,1], Jae-Sung Kwon [c,1], Taeseok Kim [a], Sung Joong Kim [a,b,*]

[a] Department of Nuclear Engineering, Hanyang University, 222 Wangsimni-ro, Seongdong-gu, Seoul 04763, Republic of Korea
[b] Institute of Nano Science and Technology, Hanyang University, 222 Wangsimni-ro, Seongdong-gu, Seoul 04763, Republic of Korea
[c] Department of Mechanical Engineering, Incheon National University, 119 Academy-ro, Yeonsu-gu, Incheon 22012, Republic of Korea

ARTICLE INFO

Article history:
Received 19 March 2018
Received in revised form 30 July 2018
Accepted 1 September 2018

Keywords:
Dry storage system
Natural circulation
PIV
Local flow field
Heat transfer characteristics

ABSTRACT

The dry storage system has received immense attention for its safe and passive cooling mechanism as a method for managing highly radioactive spent fuels that inevitably result while using nuclear power. In order to ensure the integrity of the spent fuel cladding, it is important to accurately predict heat transfer characteristics inside the system. In the study, we investigated local flow field and heat transfer characteristics of natural circulation flow at a sub-channel scale inside a downscaled dry storage system. Natural circulation flow was generated with an 8 × 8 rod bundle heaters at a power level equivalent to the decay heat of the spent fuels. The temperature and flow field were measured with thermocouples and the non-intrusive particle image velocimetry (PIV) technique by assuming axisymmetry, respectively. The results indicated that the flow at the upstream of the spacer grid rapidly accelerated at a short distance prior to entering the spacer grid due to blockage effect of a spacer grid. At the downstream, the accelerated flow is discharged as a form of jet and slowly recovers its original velocity with respect to a relatively longer decelerating region. Heat transfer characteristics inside the sub-channels were elucidated by analyzing both flow field and temperature measurements. Empirical correlations for local Nusselt number were derived by using a combination of local Grashof and local Reynolds number. The correlations indicate a clear relation between local flow field and heat transfer characteristics. The flow acceleration caused by the blockage effect of the spacer grid intensified the forced convective effect that increased the local heat transfer rate.

1. Introduction

Given the utilization of nuclear power plants to generate electricity, a key issue in management of spent nuclear fuels focuses on storing and disposing the fuels safely with a sufficient reduction in radioactivity. There are two available ways to store spent nuclear fuels, namely dry storage and wet storage. The advantages of the former over the latter are as follows: absence of the generation of radioactive water waste, absence of the need for an expensive cooling and purification system, better mobility of the spent nuclear fuels, and ease of control of radioactive leakage given an accident or attack [1,2]. Given the aforementioned advantages, dry storage constitutes a practical means for a long-term storage of spent nuclear fuels prior to permanent disposal. A dry storage system mainly consists of a metallic canister and a concrete cask [3]. The canister is backfilled with air or inert gases, such as nitrogen and helium, and accommodates spent fuel assemblies. The concrete cask encloses the canister and forms an annular flow path for ambient air between the two. The natural circulation flow of the backfill gas and the free convection flow of the ambient air dissipate the decay heat radiated from the spent fuels. Thus, the dry storage system cools spent nuclear fuels in a fully passive manner [1–4].

The efficiency of the passive cooling is directly affected by heat transfer phenomena that occur inside and outside the spent fuel assemblies. For the effective cooling of spent nuclear fuels and long-term thermal integrity of a dry storage system, it is extremely important to understand the heat transfer characteristics. Several studies investigated heat transfer characteristics inside the dry storage system numerically by using computational fluid dynamics (CFD) [5–9]. Lee investigated heat transfer inside the canister by using lump modeling of the spent fuel assemblies as a square heat source [5]. A CFD model for the prediction of heat transfer characteristics was developed by using different convective

---

* Corresponding author at: Department of Nuclear Engineering, Hanyang University, 222 Wangsimni-ro, Seongdong-gu, Seoul 04763, Republic of Korea.

E-mail address: sungjkim@hanyang.ac.kr (S.J. Kim).

[1] These authors contributed equally to this work.

https://doi.org/10.1016/j.ijheatmasstransfer.2018.09.002

boundary conditions and working fluids. The model was reasonably validated by comparing the results with experimental results in extant studies. Herranz et al. assumed a fuel assembly as a porous body and investigated the change in heat transferred from the canister based on its design, heat load, and environment temperature [8]. They concluded that heat transfer is most relevantly affected by the heat load distribution although the influence on maximum temperature is less than 4%. Shin et al. compared the heat transfer characteristics inside a canister between the various backfill gases [9]. The analysis was simplified by assuming that the system includes an 8 × 8 partial fuel assembly without any structural component such as a spacer grid. They indicated that the difference in the thermo-physical properties of the backfill gases results in dissimilar heat transfer mode.

In addition to the numerical studies, several experimental studies focused on heat transfer of natural circulation flow inside a rod bundle structure in a closed system that is similar to the canister [10–16]. Keyhani et al. [11] utilized 3 × 3 and 5 × 5 rod bundles to investigate heat transfer characteristics of natural circulation flow in a sealed metallic can. They conducted experiments with different working fluids and empirically derived correlations to predict the heat transfer characteristics for each rod in the bundle. The results indicated that heat transfer rate decreased when the location of the rod moved into the center. Heat transfer in an experimental loop containing 21 rods-bundle was examined by Hallinan et al. [12]. The natural circulation flow of water indicated that heat is transferred by mixed convection mode where forced convective effect also appears due to inertia of the flow. They developed an empirical correlation to predict the overall heat transfer characteristics by introducing the Reynolds number term. Additionally, a 10–20% enhancement of heat transfer in the system was observed when spacer grids were added to the test bundle. The enhancement was attributed to the flow area blockage effect of the spacer grids. Yoo [16] examined overall heat transfer inside the downscaled canister containing a 16 × 16 fuel assembly. Heat transfer was enhanced when the intensity of turbulence in the annular flow path increased beyond 10%. The modification of the canister surface was suggested as a practical method to intensify the turbulence.

Nevertheless, the aforementioned studies involve a few limitations in locating the local heat transfer phenomena. Although CFD simulations provide insights on the distribution of thermal and flow fields inside the system, it is difficult to simulate a prototypic dry storage system in detail. The modeling of a rod bundle structure with complex components such as spacer grids, involves high computational costs in terms of the numerical analysis. The simplification of the structures reduces the computational load although it restricts to predict the local phenomena. The results of experimental studies mainly focused only on investigating the overall heat transfer characteristics in a system scale. To the best of the author's knowledge, there is a paucity of studies that carefully investigate the local change of heat transfer characteristics inside a rod bundle structure. There are difficulties in locating local phenomena in the rod bundle structure such as spent fuel assemblies. The limited accessibility inside the sealed system and narrow width between the rods makes it difficult to monitor the temperature or flow field.

The objective of the study involves investigating local heat transfer characteristics of natural circulation flow inside sub-channels of the spent fuel assembly under dry storage. The investigation measures the local temperature and also the flow field distribution inside the sub-channels. A test section simulating the canister is established. Surface temperature of the fuel rods in each sub-channel is measured with thermocouples by assuming axisymmetry. A particle image velocimetry (PIV) technique is utilized to measure the flow field. The non-intrusive optical-based technique measures flow velocity by obtaining images of the seed particles flowing with fluid at a very short time interval and analyzing the image frames by using statistical correlation functions. The PIV is steadily utilized for the simultaneous analysis of local flow field and convective heat transfer and enables a more reliable analysis of the heat transfer phenomena [17–21]. From the temperature and flow field measurements, we investigated the local heat transfer characteristics in the sub-channels. Additionally, a local change induced by the spacer grid was elucidated by comparing the upstream and downstream data. Correlations to explain the local heat transfer characteristics and effect of spacer grid for the study are presented at the end of the study. It is expected that the results can be utilized as benchmark data to validate numerical simulations by considering the local heat transfer.

2. Experimental setup and measurement

2.1. Experimental apparatus

Fig. 1 shows the schematic diagram of the experimental setup for the study. It consists of a test section containing a rod bundle, a power level control system, a PIV measurement system, and a data acquisition system. The test section is introduced in detail in the following paragraph. The power level control system works with a 1.2 kW DC power supplier and a power distributer. It provides equivalent voltage signals to each rod in the test section. The PIV measurement system includes a 30 Hz double-frame charged coupled device (CCD) camera, a 200 mJ double-pulsed laser, a synchronizer, and a workstation computer for PIV image acquisition and analysis. The data acquisition system collects the thermocouple signals from the rod bundle in the test section.

Fig. 2(a) and (b) show the top and side-view of the experimental test section consisting of a canister, a basket, and an 8 × 8 partial fuel assembly. The height is reduced to half the height of the prototypic dry storage system [3] developed in Korea and a flow area adjusted by scaling law [22]. The test section was placed in room temperature environment, 20–22 °C. The detailed geometric dimensions of the experimental test section are given in Table 1. The wall of the aluminum canister is fabricated with transparent borosilicate glasses for PIV measurement. The basket encloses the 8 × 8 partial fuel assembly where natural circulation flow occurs. The fuel assembly contains 60 cartridge heaters simulating actual spent nuclear fuels and their decay heat. The heaters are arranged in a square lattice at a pitch of 12.85 mm. Fig. 2(c) shows the SS304 stainless steel spacer grid used to hold the fuel assembly firmly. The spacer grid for the study does not include mixing vanes in order to only consider the effect of flow area blockage in the sub-channels. The spacer grids occupy 42% of the flow cross-section in the sub-channels and are located at 640, 1,030, 1,420, and 1,808 mm from the inner bottom of the canister. At the bottom and upper part of the test section, a needle valve and a relief valve are installed, respectively. The former is to inject PIV seeds, and the latter is to prevent excessive internal pressure.

2.2. The decay heat level of spent nuclear fuel

The decay heat of actual spent nuclear fuels is simulated by providing electric power to the rod heaters through the power level control system in the study. To determine the appropriate power levels, we first calculate the decay heat level of a typical uranium dioxide (UO_2) PWR spent fuels based on a report by Ade and Gauld [23]. Hence, we consider the design constraints of a dry storage system developed in Korea, and the decay heat level of the UO_2 spent fuels with 45 GWd/MTU burn-up precooled for 1,000 days