Natural Convection Heat Transfer in the Horizontal Dry Storage System for the LWR Spent Fuel Assemblies

Motohiko NISHIMURA*, Hiroaki SHIBAZAKI**, Sadao FUJII**, and Isamu MAEKAWA*

* Nuclear Systems Division, Kawasaki Heavy Industries, Ltd.
**Kanto Technical Institute, Kawasaki Heavy Industries, Ltd.

(Received January 19, 1996), (Revised June 24, 1996)

A heat transfer and flow visualization experiment was conducted with a one-fifth scale model simulating a dry shielded canister (DSC) with 24 PWR spent fuel assemblies in order to elucidate the heat transfer characteristics and the velocity distribution for natural convection inside a DSC filled with air or water at atmospheric pressure. It was found that the average heat transfer coefficients were proportional to the one-fourth power of the Rayleigh number despite the complicated geometry inside the DSC. Flow patterns inside the DSC were visualized clearly through a digital image processing system. The velocity distributions inside the DSC were obtained quantitatively from the Particle Tracking Velocimetry. In comparison with the results of a two-dimensional thermal hydraulic analysis, computed flow patterns were similar to the experimental results and the computational temperature distributions on the sleeve surfaces agreed well with the experiments within 8%, except at the top point of the center gap. It was also found that the difference in the heat transfer coefficient was within 25% for air as the working fluid, while a satisfactory agreement was not obtained when water was the working fluid.

KEYWORDS: heat transfer, flow velocity, velocity distribution, natural convection, LWR type reactors, spent fuel assemblies, dry storage system, dry shielded canister, flow visualization, particle tracking velocimetry, thermal hydraulic analysis

I. Introduction

Horizontal dry storage system is one of the most feasible candidates for the dry storage system of the LWR spent fuel assemblies in Japan, judging from its cost and the construction interval. The major components of such a system are a horizontal storage module (HSM) made of reinforced concrete and a stainless steel dry shielded canister (DSC), as shown in Figs. 1 and 2. Each storage module is planned to house a single canister containing PWR or BWR spent fuel assemblies. The decay heat is removed by buoyancy-driven air flow. The thermal design of an HSM is categorized into the following three regions:

1. Heat transfer outside the DSC
   - Natural draft, radiation
2. Heat transfer inside the DSC
   - Convection, radiation
3. Heat transfer inside the assemblies
   - Conduction, radiation.

From the viewpoint of thermal design for the DSC, it is important to clarify the heat transfer mechanism and to establish its evaluation method in such a complicated geometry as the inside of the canister. Special attention is therefore paid to heat transfer inside the DSC.

A large number of studies on natural convection heat transfer have been performed for the simple geometries, both experimentally and numerically. Kuehn and Goldstein(1)(2), for example, have carried out experimental and theoretical studies of natural convection heat transfer in concentric and eccentric horizontal cylindrical annuli. Only a few works have, however, been directed

Fig. 1 Horizontal dry storage system arrangement
toward the study on natural convection heat transfer in an annulus with many inner rectangular cylinders. Takahashi et al. (3) carried out a 1/4 scale model experiment of the actual canister to investigate confined natural convection with thermal radiation. This study of the thermal hydraulic characteristics in the canister and the results indicated that natural convection is effective for heat transfer. Nishimura et al. (4) showed numerically that the convective heat transfer is dominant in conditions almost same condition of the prototype canister and also proposed the thermal-hydraulic similitude on the system combined natural convection with thermal radiation. However, flow patterns inside the canister were not visualized experimentally and a heat transfer correlations were not incorporated in their work.

The objectives of this study were as follows.

- to elucidate the thermal hydraulic characteristics of natural convection in the canister
- to establish an evaluation method for heat removal from the canister under combined natural convection and thermal radiation.

In this paper, a heat transfer and flow visualization experiment is described on a one-fifth scale model simulating the actual DSC system. And then, computational results using a thermal hydraulic analysis code are compared with the measurements.

II. EXPERIMENTAL APPARATUS AND PROCEDURE

The experimental apparatus is a 1/5 scale model of the actual DSC, as shown in Fig. 3. Heat inputs simulating the decay heat of the spent fuels are determined by considering the modified Rayleigh number for the actual canister, assuming that the main heat transfer in this system is caused by natural convection. It consists of 24 heated sleeves, a canister and a cooling jacket. The canister and its cooling jacket are made of Plexiglas for flow visualization.

Each heated sleeve has a square cross section with 45.2 mm in width, and 120 mm in length. An electric rubber heater is attached on the inner surface of the sleeve side wall to simulate the decay heat of the spent fuels. This heat source arrangement leads to iso-heat flux boundary conditions at the hot walls. To avoid halation, the outer surfaces of the sleeves are painted mat black with emissivity of about 0.8. The gap width between the sleeves is 8 mm, which forms a path for natural convection flow.

The canister is the cylinder with an inner diameter of 335.3 mm, 5 mm in thickness and 120 mm in axial length. Air or water coolant flows into the cooling jacket from the top wall and exits from the bottom wall after cooling the canister. Guard heaters are attached to the front and rear panels of the cooling jacket to compensate for heat loss. And the whole system is covered with thermal insulator except its bottom side. The heat loss due to axial heat transfer was 30% of the heat input for the high-power cases. Data are rejected which have more than 30% heat loss.

Chromel-alumel thermocouples are affixed to 128 points on the sleeve outer surfaces and 62 points on both inner and outer surfaces of the canister. Temperature data from these thermocouples are sent to a personal computer to be monitored and stored through data acquisition systems.

The heat input conditions are set to be in the range of modified Rayleigh number values for the actual canister; $Ra^* = 1 \times 10^7$ to $5 \times 10^8$, based on the heat flux on the outer surface of the sleeves and the inner radius of the canister. The working fluids enclosed in the canister are air and water for the low and high heat inputs, respectively. When air is the working fluid, the heat from the sleeves is removed not only by natural convection in the air, but also by thermal radiation between the surfaces of the sleeves and canister. The net heat flow by convection is therefore calculated as follows from the measured temperature distribution:

$$Q_{\text{conv}} = Q_{\text{total}} - Q_{\text{rad}},$$

where
Natural Convection Heat Transfer in the Horizontal Dry Storage System

\[ Q_{\text{conv}} : \text{Net heat flow by convection} \]
\[ Q_{\text{total}} : \text{Total heat flow at the canister wall} \]
\[ Q_{\text{total}} = \sum_{i=1}^{N} \lambda (\Delta T/\Delta r) A_C, \quad N = 13 \]  
(2)

and also
\[ Q_{\text{elec}} = Q_{\text{total}} + Q_{\text{loss}} \]  
(3)
\[ Q_{\text{elec}} : \text{Heat input of electric power} \]
\[ Q_{\text{loss}} : \text{Heat loss due to axial heat transfer} \]
\[ Q_{\text{rad}} : \text{Heat flow by thermal radiation from the sleeve surfaces facing the inner surface of the canister} \]
\[ Q_{\text{rad}} = \sum_{i,j} \sigma F_{ij} A_s (T_i^4 - T_j^4), \quad N_i = 120, \quad N_j = 160. \]  
(4)

In the experiment, the rate of \( Q_{\text{conv}} \) was from 35% to 45% of \( Q_{\text{elec}} \), and \( Q_{\text{rad}} \) was evaluated with the measured axial temperature distributions in the canister. Consequently, \( Q_{\text{loss}} \) was close to \( (Q_{\text{elec}} - Q_{\text{total}}) \).

Measurements were made in the steady state as recognized through monitoring the temperature history and heat flows in the canister. It took 8 to 12 h for the experiment to reach the steady state. The temperature values used are averages from a series of five time points for each measurement location.

The average heat transfer coefficient, \( h \), is calculated as follows:
\[ h = q_s/(T_S - T_C), \]  
(5)
where \( q_s \) means heat flux on the sleeve surfaces, \( Q_{\text{conv}}/A_{\text{sleeve}} \). \( A_{\text{sleeve}} \) is a total area of the sleeve side walls, \( T_S \) is the average temperature of the sleeve surfaces along the gap in the vertical center line of the canister, and \( T_C \) the average temperature of the inner surface of the canister. The average Nusselt number is then defined by
\[ Nu = h R_C / \lambda, \]  
(6)
where \( \lambda \) is the thermal conductivity of the enclosed fluid and \( R_C \) the inner radius of the canister.

III. FLOW VISUALIZATION TECHNIQUE

Zinc stearate and polystyrene were selected as tracing materials for the working fluids of air and water respectively. The zinc stearate has a diameter of 2 to 8 \( \mu \)m with low density. Talcum powder was also tried as a tracing material. The zinc stearate, however, was selected finally for the air, because of its light scattering ability and the long suspension time in the gas. The polystyrene particle is about 10 \( \mu \)m in diameter and its specific weight is about 1.06. These tracer particles are injected into the working fluid after the steady state has been reached. It takes 3 to 5 min to recover from the perturbation of the flow field caused by the injection. For this reason, photographs of the flow fields are taken in 10 min after injection. The measurement error of the fluid velocity due to the density difference between tracer particles and the working fluid is negligible small in the experiment.

The measurement system for flow visualization consists of a strobe light source, a high-resolution camera with broad dynamic range, a laser videodisk recorder, and an image processor. The strobe slit light is mounted over the apparatus to light up the vertical cross section. It is synchronized with the high-resolution camera which takes 30 frames per second. Recognition of micron-size particles is achieved by the camera which emphasizes luminosity contrast of the photographic objects.

Simultaneous visualization of the flow pattern and the temperature distribution is made possible by suspending in the water with 3% micro-capsulated thermally sensitive liquid crystals.

A particle tracking velocimetry (PTV) which is installed in the image processor calculates velocity distributions from a series of three pictures taken at certain time intervals. Velocity vectors are obtained for arbitrary positions within the frame, so that the vectors are interpolated to certain grid points by means of a Gaussian window. The velocity data presented here are the average values over 6 pictures taken in 4 s.

IV. NUMERICAL ANALYSIS

A two-dimensional thermal hydraulic analysis is performed to validate the applicability of the code named ZEPHYRUS, and to gain further understanding of the heat removal characteristics of the canister by additional information through the analysis. Outline of the code is shown in Table 1.

It is assumed that the thermal hydraulic field is uniform in the axial direction and symmetrical across to the vertical centerline of the canister owing to geometry. The computation thus was performed in two dimensions on a half sector model consisting of 3,200 computational mesh points (see Fig. 4). The inner surface of the canister is modeled by a stepwise mesh arrangement using the porous body model. The computation considers both convection and radiation heat transfers.

The boundary conditions are as follows:

Momentum equations.

Gap between the sleeves: Friction factor of 96/Re
Sleeve surfaces facing to canister: Non-slip condition
Inner wall of the canister: Free-slip condition

Energy equation.

Heat source in the sleeves: Heat generation rate is specified in sleeve walls based on \( (Q_{\text{elec}} - Q_{\text{loss}}) \)
Canister temperature: Experimental value is specified

Heat transfer between the structures and the fluid: No correlation used, (that is conduction, \( Nu=1 \))
**V. RESULTS AND DISCUSSION**

1. Heat Transfer Characteristics

The experimental and computational heat transfer characteristics of the canister are shown in Fig. 5. The average Nusselt numbers are evaluated using Eq.(6) and the Rayleigh numbers are defined by

$$Ra = g\beta(T_S - T_C)R_3^3 Pr/\nu^2.$$  \hspace{1cm} (7)

The computational results of heat transfer coefficient are in fairly good agreement with experimental results for air as the working fluid. The difference in the heat transfer coefficient is within 25%. However, a satisfactory agreement is not obtained when water is the working fluid. The type of coolants outside the canister has no effect on the heat transfer characteristics inside the canister. It is well known that the heat transfer coefficients are proportional to the 1/4 power of the Rayleigh number for laminar natural convection in a simple geom-

---

### Table 1 Outline of ZEPHYRUS\(^{(9)}\)

<table>
<thead>
<tr>
<th>1. Basic formulation</th>
<th>Governing Equations: (u, v, w, p(\rho), h, c, T_w)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2. Numerical method</td>
<td>Algorithm: SIMPLER(^{(10)}), SIMPLEST-ANL(^{(11)})</td>
</tr>
<tr>
<td></td>
<td>Linearization: Iterative method</td>
</tr>
<tr>
<td></td>
<td>Matrix solver: MICCG(^{(12)}) (for p eq.)</td>
</tr>
<tr>
<td></td>
<td>Point SOR (for all eqs.)</td>
</tr>
<tr>
<td>3. Discretization scheme</td>
<td>Convection term: 1st-order skew upwind</td>
</tr>
<tr>
<td></td>
<td>Diffusion term: 2nd-order central differential</td>
</tr>
<tr>
<td></td>
<td>Time integration: Implicit euler</td>
</tr>
<tr>
<td>4. Physical models</td>
<td>Porous body model: COMMIX(^{(13)}) type, Dynamic model(^{(14)})</td>
</tr>
<tr>
<td></td>
<td>Flow resistance model: Distributed flow resistance model based on absolute velocity</td>
</tr>
<tr>
<td></td>
<td>Heat transfer model: Correlation equations</td>
</tr>
<tr>
<td></td>
<td>Thermal radiation model: Multi-pass model solving radiosity equations(^{(15)})</td>
</tr>
<tr>
<td></td>
<td>(for structure surfaces)</td>
</tr>
<tr>
<td></td>
<td>Turbulence model: (k-\varepsilon-\theta^2) equations</td>
</tr>
</tbody>
</table>

* The underlines denote the options adopted in this computation.

---

![Fig. 4 Computational mesh system (40×80)](image)

![Fig. 5 Heat transfer characteristics](image)
metry. In spite of the complex geometry in this study, the average heat transfer coefficients are approximately correlated by an equation of the same form: \( N_u \propto Ra^{1/4} \) in the range within \( 10^6 < Ra < 10^8 \), consequently. This interesting result implies that the flow regime in the canister is laminar natural convection. When the working fluid is air, the heat transfer coefficients are proportional to the about 0.27 power of the Rayleigh numbers. While the heat transfer coefficient is proportional to the about 0.19 power of the Rayleigh numbers for the correlation including both kinds of working fluids: air and water. For the data obtained for water as the working fluid, the accuracy deteriorates because of the small temperature differences.

**Figure 6** gives experimental and computational temperature distributions on the sleeve surfaces. The computations agree well with the experiments within 8%, except at the top point of the center gap. This agreement also offers circumstantial evidence in favor of a laminar flow regime in the canister. One of the causes of the discrepancy between computation and experiment is thought to be a three-dimensional effect arising from axial heat transfer inside the canister.

The peak temperature is located at the second sleeve from the top along the vertical center line gap. Similarly, the temperature profile for every vertical gap has maximum temperature point in its upper part. This fact indicates that the heat transfer by convection is dominant in vertical direction inside the canister. The temperature differences among the vertical gaps falls with height, due to the buoyancy effect which accompanies horizontal flows.

### 2. Flow Field Behavior

**Figure 7** illustrates typical overall flow patterns inside the canister as observed by eye. The flow pattern has three main characteristics: global circulation, local circulation and the upflow through the horizontal...
gaps. In addition, in the condition of $Ra \geq 4 \times 10^7$, an almost stagnant and thermally stratified region is observed clearly in the lower one third of the canister (see Fig. 7(b)). The flow path of the global circulation driven by buoyancy consists of upward flows heated through each vertical gap between the sleeves, and downward flows of cooled fluid along the inner surface of the canister. The local circulation takes place between the sleeves and the canister walls caused shear forces as a result of generated global circulation. Horizontal flows cross from the peripheral region of the sleeves to the center gap, resulting in the temperature equalization among the vertical gaps mentioned above (see Fig. 6).

Flow patterns of local circulation measured with the PTV are shown in Fig. 8. In the region shown in Fig. 8, a counter-clockwise circulation is observed. The condition $Ra = 4.8 \times 10^6$ of air shows vectors of tracer particles near the center of the flow circulation, while the condition $Ra = 4 \times 10^7$ for water shows only a few tracer particles near the same region. In the latter case, it is difficult for the tracer particles to flow into vicinity region of the circulation center, because working fluid is almost stagnant there. This flow pattern can be explained as follows. Water has Prandtl number about ten times larger than air. The potential for heat penetration from heated or cooled walls into the bulk of the fluid falls with higher Prandtl number. This will cause steeper temperature gradient and thinner boundary layers with a higher Prandtl number, resulting in a stagnant region in the bulk of the fluid.

Figure 9 shows the velocity fields at the top of the gap at the vertical center. The upward velocity distribution in the transverse direction of the gap has not been measured thoroughly. The discretely measured velocity in the gap is laying in the range from 1.4 to 2.2 mm/s, and the computation gives the average gap velocity of 1.4 mm/s.

Computed flow patterns and temperature distributions are given in Fig. 10. The overall flow pattern is similar to the experimental result shown in Fig. 7 (see I, II and III in Fig. 10). The temperature distribution in the water case shows a stratified lower part of the one third of the height of the canister; this was also observed experimentally with the thermocouples and the microcapsulated thermally sensitive liquid crystals. The computation also gives the discrepancy of the flow pattern near local circulation centers between the two working fluids (see Fig. 8). The computational result thus agrees well qualitatively with the experiment.

Quantitative comparisons of velocity distribution between the experiment and the computation are shown in Figs. 11(a) and (b). The experimental values measured with the PTV are obtained through averaging over two seconds and by interpolating for certain points using a Gaussian window. The trends of the velocity distributions are almost similar in both results. The compu-
Natural Convection Heat Transfer in the Horizontal Dry Storage System

Fig. 10 Computational results of flow and temperature fields

(a) $Ra=9.3 \times 10^6$, Fluid: Air

(b) $Ra=4.0 \times 10^7$, Fluid: Water

An interval of isothermal temperature: 5°C

Fig. 11 Comparison of velocity distribution in horizontal and vertical directions ($Ra=4.0 \times 10^7$, Fluid: Water)

(a) Horizontal direction

(b) Vertical direction
tion does not show quantitative agreement of the absolute values with experiment, because the averaging time span of the experimental value is not long enough compared with that of natural circulation.

VI. CONCLUSIONS

A heat transfer and flow visualization experiment was conducted with a 1/5 scale model simulating a DSC with 24 PWR spent fuel assemblies. The heat transfer characteristics and the velocity distribution for natural convection inside the DSC. Results were obtained for both air and water as the working fluids at an atmospheric pressure. The modified Rayleigh number based on the inner radius of the DSC varied from $3 \times 10^7$ to $2 \times 10^8$.

It was found that the overall heat transfer coefficients were proportional to the one-fourth power of the Rayleigh number. The present result also coincides with another experiment under the high temperature condition ($>200^\circ C$), of which geometry is the same but a one-fourth scale model using a mixture gas. The flow patterns inside the DSC were clearly visualized using a digital image processing system. The velocity distributions of the fluid in the DSC were obtained using a PTV.

In comparison with the results of a two-dimensional thermal hydraulic analysis, computed flow patterns were similar to the experimental results and the computational temperature distributions on the sleeve surfaces agreed well with the experiments within 8%, except at the top point of the center gap. It was also found that the difference in the heat transfer coefficient was within 25% for air as the working fluid, while a satisfactory agreement was not obtained when water was the working fluid.

[NOMENCLATURE]

$A_{Cl}$: Area fraction where a couple of thermocouples concerns on inner and outer surfaces of the canister

$A_{Si}$: Area of one side of the sleeve outer surface

$c$: Mass fraction of chemical species (mass concentration)

$F_{ij}$: Radiation exchange factor

$g$: Gravitational acceleration

$h$: Specific enthalpy

$p$: Pressure

$Pr$: Prandtl number

$\Delta r$: Thickness of the canister wall

$Ra^*$: Modified Rayleigh number ($=g\beta qsR_c^2Pr/\lambda u^2$)

$\Delta T$: Temperature difference between inner and outer surfaces of the canister

$T_i$: Outer surface temperature of a side of the sleeve facing to the inner surface of the canister

$T_o$: Inner surface temperature of the canister measured by the thermocouple affixed opposite to i surface of the sleeve

$u, v, w$: Velocity component respectively in the $x, y, z$ directions

$\beta$: Volumetric expansion coefficient

$\epsilon$: Dissipation rate of turbulence kinetic energy

$\theta$: Temperature fluctuation

$\lambda$: Thermal conductivity

$\nu$: Kinematic viscosity

$\rho$: Specific weight

$s$: Stefan-Bolzmann constant

ACKNOWLEDGMENTS

The authors wish to express their gratitude to Assistant Professor K. Nishino of Yokohama National University for his instructive comments and useful suggestions.

REFERENCES


(2) ibid.: Trans. ASME., J. Heat Transfer, 100, 635-640 (1978).


JOURNAL OF NUCLEAR SCIENCE AND TECHNOLOGY