Two-Dimensional CFD Simulations of a Square 8x8 Heater Rod Array in and Isothermal Enclosure Filled with Rarified Air

Miles Greiner, Ph.D., Pablo Araya, N.R. Chalasani, Ph.D.
Mechanical Engineering Department, Mail Stop 312, University of Nevada, Reno, Reno, NV 89557, greiner@unr.edu

Jie Li, Ph.D. and Yung Liu, Sc.D.
Argonne National Laboratory, 9700 Cass Ave Argonne, Illinois, 60439, yyliu@anl.gov

In this work a two-dimensional computational fluid dynamics model is constructed of an existing experiment that consists of a square 8x8 array of heater rods within a square cross section pressure vessel filled with air. This model includes heat generation and conduction within the rods, natural convection and radiation heat transfer across the air-filled region between the rods and enclosure, and the effective thermal resistance at the gas/solid interfaces that is significant at low pressures when the gas is rarified. This configuration is relevant to the vacuum drying process that is used when used nuclear fuel is transferred from underwater to dry storage. Simulations at an air pressure of 500 Pa, an enclosure temperature of 325°C and an array heat generation rate of 500W indicate the rarified-gas thermal-resistance increases the centermost rod temperature by 7°C compared to simulations that do not include that effect. This is roughly equal to the random differences between measured and simulated maximum rod measurements from an earlier study conducted at higher gas pressures. In order to acquire data that can be used to confidently benchmark rarified gas simulations, the experiment must be operated at conditions that cause the rarified gas model to predict maximum temperatures that are more than 7°C hotter than continuum gas predictions. This may be achieved by using lower wall temperatures and or higher array heat generation rates, which will be the subject of future work.

I. INTRODUCTION

Used light water reactor fuel rods consist primarily of zircaloy cladding tubes that contain highly radioactive fuel pellets as well as high pressure fission-product and fill gases [1]. The fuel rods are held in a square array by periodic spacer plates. Boiling water reactor (BWR) assemblies consist of 7x7 to 9x9 rod arrays surrounded by a solid channel. Pressurized water reactor assemblies are 14x14 to 18x18 arrays, but do not have channels.

After being discharged from reactors, these assemblies are stored underwater while their radioactivity and heat generation rate decrease [2]. After sufficient time, typically five years or more, a canister with an internal basket is placed in a transfer cask and lowered into the pool. The canister is then loaded with fuel assemblies, covered, and lifted out of the pool. Helium or another non-oxidizing gas is forced into the canister through a port near its top while water flows out through a tube that reaches nearly to the cask bottom [3]. Small amounts of water may remain at the bottom of the canister and in crevices of the canister and cladding surfaces after draining. Essentially all moisture must be removed from the canister before it is sealed to prevent corrosion of the fuel cladding and cask materials [4]. After drying, the canister is filled with helium to pressures between 3 and 7 atm (406 to 711 kPa) and sealed either by welding or bolted closure. It is then placed in other packaging for onsite dry cask storage or offsite transport.

With the absence of a defined used-fuel disposal and/or reprocessing path, it is crucial to assure the safety of long-term dry cask storage systems [5]. Federal regulations (10CFR72) require that these systems insure that external doses are below certain limits, and that the fuel configuration remains subcritical, confined and contained, and retrievable. The cladding is the primary confinement barrier for the used fuel pellets and fission gas. Its integrity must be protected to assure that, after decades in storage, the assemblies can be safely transferred to other packages, and/or transported to other locations.

Federal regulations (10CFR71) also require that the transport package performance be analyzed under normal conditions of transport (which include a 0.3-m drop) and hypothetical accident conditions (which include a 9-m drop). If the cladding integrity is compromised there is a risk that the fuel will not be in its “as analyzed” configuration after these drop events. Adequate ductility of the cladding must therefore be maintained. Radial hydride formation within the cladding has the potential to radically reduce cladding ductility and its suitability for long term storage or transport.

During all post-reactor drying, transfer, storage and transport operations the fuel cladding must be kept below certain temperature limits to avoid (a) dissolution of circumferential hydrides that exist in the cladding and (b) high gas pressures within the tubes, which leads to high
cladding hoop stress [6]. If these hydrides dissolve and the hoop stresses become large, then as the heat generation of the used fuel decreases during long-term storage radial hydrides may form and cause the cladding to become brittle. Drying operations may be the most likely events to cause the fuel temperature to exceed temperature limits. This is because drying is the first operation when the fuel is removed from water and placed in a gas-filled environment, and the fuel heat generation is still relatively high.

Nuclear Regulatory Commission Interim Staff Guidance-11, Revision 3 (ISG-11) [6] specifies conditions that are intended to prevent radial hydride formation. For example, the maximum calculated fuel cladding temperature must remain below 400°C for normal conditions of storage and short term loading conditions (e.g., drying, backfilling with inert gas, and transfer of the cask to the storage pad). For low burnup fuel, a higher short-term temperature limit may be used, provided that the best estimate cladding hoop stress is less than 90 MPa for the temperature limit proposed. During loading operations, repeated temperature cycling is allowed, but is limited to less than 10 cycles with cladding temperature variations of more than 65°C. For off-normal and accident conditions, the maximum cladding temperature should not exceed 570°C, a limit based on creep (stress) rupture consideration. Until further guidance is developed, high burnup fuel will be handled on a case-by-case basis [7-11].

Two methods are currently used by industry for moisture removal from canisters, vacuum drying or forced helium dehydration [3, 6]. In vacuum drying, the canister is evacuated to pressures as low as 67 Pa to promote evaporation and water removal [4]. Several cycles of evacuation and refill may be necessary before operators can demonstrate that the canister is able to meet the drying technical specification of maintaining a low pressure of 400 Pa (3 Torr) for 30 minutes [3, 11].

At the low pressures and gas densities associated with vacuum drying, buoyancy-induced gas motion and natural convection heat transfer from the fuel to the canister surfaces are essentially eliminated. While the gas thermal conductivity is nearly the same at these pressures as it is at atmospheric conditions, the gas is rarified to the extent that there is a temperature difference (or temperature-jump) between the heated cladding and the gas in contact with it [4, 12-15]. This surface-to-media temperature-jump is essentially zero at moderate pressures but acts as a thermal resistance between the surfaces and gas at low pressures. These resistances increase the cladding temperature compared to atmospheric pressure conditions. This thermal resistance and the lack of natural convection caused by low pressure may lead to higher cladding temperatures during vacuum drying than during storage conditions for the same fuel heat generation rate.

Forced helium dehydration is used for drying canisters containing high-burnup fuel [3]. In that process, helium is forced into the canister through a port near its top and withdrawn through a tube that reaches to the cask bottom. Moisture is removed from the helium by condensing, demoisturizing, and preheating the gas outside the canister. In some cases cooling water is also circulated in the gap between the canister and transfer cask. The gas pressure during helium dehydration is maintained at roughly the same level as that used during storage. As a result the same natural convection and minimal temperature-jump thermal resistance are expected as in storage. However, gas demoisturizing and cooling equipment are required for forced helium dehydration, which are not needed for vacuum drying.

Cladding temperatures and hoop stresses during drying are calculated using experimentally-benchmark whole-package computational fluid mechanics (CFD) simulations in which the fuel and basket are replaced by a region with an effective thermal conductivity and porosity [3, 16]. This allows prediction of the maximum fuel heat generation rate that can be transferred without exceeding the temperature and hoop stress limits. It also helps determine which fuel may be vacuum-dried, and for which fuel the more complex forced helium dehydration process must be used.

The whole-package computational methods have been validated [17] against measurements performed in an actual evacuated storage package [18]. Currently, these effective properties are calculated without regard to the rarified-gas temperature-jump thermal resistance. However, the fuel heat generation in the tests used to validate the current methods was moderately low. The effect of the rarified-gas thermal resistance on peak cladding temperatures increases with generation rate.

**Current Work** The long term objective of the current research program is to develop and experimentally-benchmark CFD models of the vacuum drying process that include the effect of the rarified-gas thermal resistance. This work will eventually employ an existing experimental apparatus that consists of an 8x8 array of heater rods within an aluminum pressure vessel (Fig. 1). This apparatus simulates a region of a BWR assembly within its channel and between consecutive spacer plates. Experiments will be performed to acquire rod temperatures for a range of rod heat generation rates, wall temperatures and helium gas pressures. Low pressures relevant to the vacuum drying process will be examined.

In the current paper two-dimensional CFD simulations are performed to model the apparatus filled with air at 500 Pa. The simulations include heat generation and conduction within the rods, natural convection and radiation heat transfer across the air-filled region between the rods and enclosure, and the rarified gas thermal resistance at the solid/gas interfaces. The
objective is to quantify how much this thermal resistance increases the maximum or peak rod temperature compared to simulations that do not include that effect. These results will be used to determine the experimental conditions under which this effect is sufficiently large that it can be differentiated from random variations in the experimental temperature measurements. In the future the test facility will be used under these conditions to benchmark simulations of drying operations.

II. NOMENCLATURE

1 ATM Atmospheric pressure model
500 Pa Low pressure model which assumes a continuum flow regime
500 Pa-Tj Low pressure model which temperature jump on all surfaces
BWR Boiling Water Reactor
CFD Computational Fluid Dynamics
cp Specific Heat [J/kgK]
d Diameter of the molecules of cover gas [m]
g Acceleration due to gravity [m/s2]
Kn Knudsen Number [-]
k Boltzmann constant, 1.38 × 10^-23 [m2kg/s2K]
Lc Characteristic Length [m]
m molecular mass of the gas [kg]
P Local pressure [Pa]
Pr Prandtl Number (cpc/μk)
PWR Pressurized Water Reactor
Q Total array heat load [W]
q” Heat flux [W/m2]
Tg Temperature of the gas near the wall [°C]
TMAX Maximum temperature within domain [°C]
Tw Temperature of the wall [°C]
T Local temperature [°C]
α Thermal accommodation coefficient [-]
ΔTMAX Tmax-Tw; Maximum rod to enclosure wall temperature difference [°C]
e Surface emissivity [-]
γ Specific heat ratio [-]
κ thermal conductivity of the gas [W/MK]
λ Mean free path [m]
μ Stress viscosity of the gas [kg/ms]
ζr Temperature Jump Coefficient (TJC) [-]

III. EXPERIMENTAL APPARATUS

Figure 1 shows the disassembled test facility that models the region of a used BWR fuel assembly between consecutive spacer plates and within its channel. It was originally constructed to model the thermal conditions during used fuel transport and storage [19, 20]. On the left side of Fig. 1 is an 8x8 square array of rods held by spacer plates at both ends. The rods contain electric heaters and internal thermocouples. The array is placed in a square cross-section anodized-aluminum pressure vessel (right side of Fig. 1). The spacer plates and vessel walls also contain thermocouples. Stainless steel endplates (not shown) are bolted to both ends of the enclosure, and sealed using high-temperature O-rings. The heater power cables and thermocouple lead wires are connected to high-temperature feedthroughs in the endplates. A computer data acquisition system and a power supply are connected to the outer terminals of the feedthroughs to record temperatures and power the heaters. Steady-state rod and surface temperature measurements were made in seventy-two experiments under the following conditions: (a) helium or nitrogen fill gases at pressures of 1 to 3 atm, (b) rods in the vertical (storage) and horizontal (transport) orientation, (c) array heat generation rates of 100 to 500 W, and (d) different thicknesses of insulation surrounding the system (to increase the aluminum enclosure temperature to up to 280°C).

Figure 2 is a schematic cross-section of the experimental 8x8 heater rod array within the interior surface of the square cross section aluminum enclosure. Each rod is 1.1 cm in diameter and has a 60.9-cm heated length. Individual heater rods are made up of compressed MgO core surrounded by a 0.7-mm-thick Incoloy sheath. The heaters are arranged in a square pattern with center-to-center pitch spacing of 1.46 cm. The enclosure interior surface is a square, 12 ± 0.25 cm on each side. The array is centered within the aluminum vacuum chamber. As a result the minimum spacing between the rods and the wall is 3.35 mm, while the minimum spacing between adjacent rods is 3.6 mm.

The 60.9-cm heated rod length is shorter than the typical 3.6-m active length of a fuel assembly. As mentioned before, the apparatus is intended to be representative of a region between consecutive grid spacers [1]. The surface emissivity of the rods and chamber walls are 0.8 (specified by the manufacturer) and 0.5 for the aluminum walls [21].

Figure 3a shows the apparatus wrapped in insulation in the vertical orientation. Figure 3b shows heater surface temperature contours calculated from a three-dimensional CFD simulation of the experimental apparatus in the vertical orientation [20]. Half of the heaters are removed to show the hottest rods. This calculation was performed using the Fluent CFD package with the enclosure filled with nitrogen gas at 3 atm. The simulations include heat...
generation and conduction within the rods, and natural convection and radiation heat transfer across the gas. Figure 3b shows buoyancy-induced gas motion causes the highest temperatures to be above the array center. Simulations with helium (not shown), which has a higher thermal conductivity than nitrogen and so are not as affected by buoyancy, show the maximum temperature is closer to the rod mid-height. For the 72 experiments, 95% of the simulated temperature differences between the hottest rod and the aluminum enclosure were within 7°C of the measured data.

In the current work we wish to determine if the apparatus described in this section can be used to benchmark rarified-gas simulations under conditions that are relevant to vacuum drying. In order to do that with a high degree of certainty, the experiment must be operated under conditions that cause the maximum temperature predicted by rarified gas simulations to be at least 7°C hotter than simulations that do not include this effect.

IV. RARIFIED GAS HEAT TRANSFER

Under normal atmospheric conditions, gas molecules confined within a moderate-sized volume experience many collisions with each other between interactions with the walls (4, 12-15). The mean free path between molecular collisions is:

\[
\lambda = \frac{kT}{\sqrt{2\pi P d^2}} \quad (1)
\]

where \( k \) is the Boltzmann constant, \( P \) is the local pressure, \( T \) is the local temperature, and \( d \) is the diameter of the molecules. From equation (1), it is clear that the mean free path increases as the pressure decreases and/or the temperature increases.

Knudsen Number (Kn) is the ratio of the mean free path to the characteristic length \( L_C \) of the region occupied by the gas:

\[
Kn = \frac{\lambda}{L_C} \quad (2)
\]

We note that \( L_C \) is easily defined in simple enclosures such as parallel plates, spheres and cylinders. However, in complex geometries such Figure 2 the characteristic length is not easily determined.
If $\lambda \ll L_C$, a molecule will experience “many” molecular collisions between interactions with the walls, and the molecules at any location reach equilibrium with each other. As a result when $Kn \ll 1$, the gas may be treated as a continuum. When $Kn < 0.01$, the Navier-Stokes and Convective Energy equations accurately model momentum and energy transport within a gas [13]. Moreover, at the interface between the gas and solid surfaces, the gas and wall temperatures and their velocities are effectively the same, that is $T_g = T_w$, and $V_g = V_w$ (no-slip boundary condition).

Equations (1) and (2) show that the molecular mean free path and the Knudsen number increase as the gas pressure decreases. If $Kn$ is sufficiently large then a molecule experiences only a few molecular interaction between wall collisions. Under those conditions, the system exhibits characteristics of a coarse molecular structure, and the gas is considered rarified. When $Kn > 0.1$ the Boltzmann kinetic equation must be applied to accurately model the gas, and its numerical solution is computationally intensive [13].

For $0.01 < Kn < 0.1$ the gas is considered to be at a level of slight rarefaction [14]. In such cases the gas tends to behave as a continuum in regions away from the walls. However, a molecule that comes into contact with a wall does not meet other molecules enough times to reach equilibrium with them in the vicinity of the wall [15]. Therefore there can be an abrupt change of temperature and speed from the surface to the gas, that is $T_g \neq T_w$, and $V_g \neq V_w$. This is known as temperature-jump or slip-flow. As a result, for $0.01 < Kn < 0.1$, the Navier-Stokes and Convective Energy equations accurately model momentum and energy transport away from the walls, but gas rarefaction must be taken into account at the walls using “temperature-jump” and “velocity-slip” boundary condition [13, 14]. Due to the difference in the gas and surface velocity, this condition is known as the slip-flow regime.

As an example, Fig. 4 shows the dependence of the maximum Knudsen number on pressure for experimental apparatus filled with air. This plot is based on Equations (1) and (2), and an equivalent spherical diameter for air of $d = 3.72$ Angstrom [12]. In order to maximum the Knudsen number this plot uses the maximum temperature at which the apparatus can operate $T = 325°C$ (based on the O-ring seal temperature limit), and the minimum surface-to-surface spacing in the apparatus $L_C = 3.35$ mm (between the outermost rods and the enclosure walls, Fig. 2). Results for nitrogen ($N_2$), argon (Ar) and helium (He) gas are included using their molecular diameters. The effective diameter of He is much smaller than those of the other gases. At $P = 500$ Pa, $Kn = 0.008$ for air, Ar and $N_2$, which is at the upper end of the continuum regime, while $Kn = 0.022$ for He, which is within the slip-flow regime.

The local temperature difference or jump between the gas and wall is determined from:

$$T_g - T_w = \zeta T \frac{\mu (2kT_w)}{m} \frac{q''}{\kappa}.$$  \hspace{1cm} (3)

In this expression $\mu$ is the gas dynamic viscosity, $m$ is the mass of a gas molecule, and $q''$ is the local heat flux normal to the surface and directed into the wall. If the heat flux is to the wall then the gas temperature is greater than the wall temperature, and vice versa. The temperature-jump therefore acts as a thermal resistance between the gas and surface. We note that the heat flux $q''$ is the component transported by conduction and convection within the surrounding gas, and does not include the component transported by radiation to other surfaces.

The Temperature Jump Coefficient $\zeta_T$ in Equation (3) is determined by applying the Boltzmann Equation to the Knudsen Layer [13], and can be calculated as

$$\zeta_T = \frac{2 - \alpha}{\alpha \left( \frac{\gamma}{\gamma + 1} \right) Pr}, \quad Pr = \frac{\mu}{\kappa} c_p .$$ \hspace{1cm} (4)

In this expression $\alpha$ is the gas/surface accommodation coefficient, $\gamma$ is the gas specific heat ratio, $Pr$ is the gas Prandtl number, $\kappa$ is the thermal conductivity of the gas, and $c_p$ is the specific heat at constant pressure. For air at $T = 300$ K in contact with aluminum, $\alpha = 0.87-0.95$ for a polished surface, and $\alpha = 0.95-0.97$ for a machined surface [17]. Using the values $\alpha = 0.95$, $\gamma = 1.4$ and $Pr = 0.71$ equation (4) give $\zeta_T = 1.609$.

In the experimental apparatus, the heat generation rate within in all the rods is essentially the same, so the average heat flux rate leaving each rod is also the same. For example, for an assembly of 64 rods with a total heat generation rate of 500 W, and heater length and diameter of 60.9 cm and 1.1 cm, respectively, the average heat flux leaving the rod by conduction, convection and radiation to its surroundings is 371 W/m$^2$. For the square enclosure
with sides of length 12 cm, the average heat flux to it is 1710 W/m².

However, the local heat flux to or from any point on the rod or enclosure surface depends on the local surface temperature and the gas and other surface temperatures surrounding it. For the four rods at the center of Fig. 2, the heat flux is nearly zero on the surfaces that face the center of the assembly (due to symmetry), but larger than the average on the surface regions facing away from the center. For the “ring” of 12 rods that surround the center four rods, there is net heat flux to the rod surfaces that face the array center, while the net flux on the surfaces facing the outer regions of the array is away from the rods. The total amount of heat transferred to the inner surfaces and away from the outer surfaces of each ring of rods increases as the distance between the ring and array center increases. Due to the direction of heat flux to and from each rod, there are temperature-jumps on the inner and outer surfaces that compound the effects of gas rarification on the temperature difference between the center rods and the enclosure.

V. COMPUTATIONAL METHODS

This section describes the computational methods used to calculate the rod temperatures when the experimental apparatus is filled with air, for a range of array heat generation rates and gas pressures. For assembly heat generation rates of 100 to 500 W and a heated length of 60.9 cm, the axial heat generation rates are between 164 W/m to 820 W/m. For a typical BWR with a length of 3.6 m and a peaking factor of 1.25 [1], these axial heat generation rates correspond to the maximum values within assemblies that generate 470 to 2300 W. The enclosure temperature used in this paper is 325°C, which is the highest temperature that can be used in the experiment based on the enclosure O-ring seal temperature limit.

Dashed lines in Fig. 2 surround four heater rods and a portion of the enclosure surface at the bottom right of the apparatus. Figure 5 shows the finite volume grid of this region used in this paper. The full grid, which includes all 64 rods and the gas between the rods and the interior surface of the enclosure, consists of 46,592 elements. The current authors used the meshing scheme used to create this grid for two-dimensional CFD simulations of a 7x7 array with nitrogen at atmospheric pressure, and showed the results were independent of further mesh refinements [22]. Future simulations with the current geometry at low pressures will be run with different grid refinements to confirm grid independence.

The simulations presented in this study were performed using the FLUENT 6.3.26 CFD package. A steady state solver with a second-order upwind scheme was used to solve the mass, momentum and energy equations. Pressure-velocity coupling was solved using the SIMPLE method. Radiative heat transfer was solved for gray diffuse surfaces using the discrete ordinates method with a second-order upwind scheme. The governing equations were solved using double precision. Buoyancy-induced flow is generated by adding the acceleration due to gravity, g = 9.8 m/s² in the –y direction, as shown in Fig. 5. The ideal gas law was used to determine the pressure and temperature dependent air density.

Vacuum drying is normally conducted with the fuel rods in the vertical orientation. Calculation of natural convection among vertical rods requires the use of a three-dimensional domain. In the preliminary simulations presented in the current work the rods are horizontal to allow the use of a two dimensional grid. Future work will consider three dimensional simulations of vertical fuel assemblies.

In this work the maximum rod temperature is determined for total heater array heat generation rates of Q = 100 to 500 W. Four different gas heat transfer models were employed. The first two were performed for a pressure of 500 Pa, which is relevant to vacuum drying. In the first of these the rarified gas temperature-jump thermal resistance is included by implementing a finite conductance between the gas and solid regions. This is referred to as the 500 Pa-T model. To implement the temperature jump in the CFD simulation, equation (3) is written in terms of a temperature-jump conductance,

\[ q'' = \kappa_{\text{jump}} \left( T_g - T_w \right) \]  \hspace{1cm} (5)

where
Equation (6) shows $\kappa_{jum}$ is a function of the local wall temperature. However, a temperature-dependent contact conductance was not easily implemented in the Fluent code. In this preliminary work, an iterative approach was used for each heat load. A simulation was performed to determine the area-average rod surface temperature. This value of $T_w$ was then used in equation (6) to find the conductance at all gas/solid interfaces. This process was repeated until $T_w$ converged. Future simulations will implement jump conductances that are dependent on local wall temperatures.

In the second $P = 500$ Pa model, the gas is modeled as a continuum and the rarified gas thermal resistance is ignored, $T_g - T_w = 0$. This continuum model is referred to as the $500$ Pa-C model, and comparing it to the $500$ Pa-Tj model quantifies the net effect of the rarified gas thermal resistance. In the third model the gas pressure is $1$ atm ($101.3$ kPa). At this pressure the rarified gas temperature jump is essentially zero. Moreover, the gas density is significantly higher than it is for the $500$ Pa simulations and so buoyancy induced gas motion and natural convection heat transfer are more active. This model is referred to as $1$ Atm-C, and comparison of it to the $500$ Pa-Tj model quantified the effect of gas pressure.

The fourth and final model assumes all gas is removed from the system so that all heat transfer from the heater rods to the enclosure is by radiation. These simulations are conducted with a gas conductivity of $1 \times 10^{-8}$ W/mK. This model is referred to as Hard Vacuum model; the difference between it and the $500$ Pa-Tj model quantifies the effect of conduction and convection through the gas.

\[
\kappa_{jum} = \frac{\kappa}{\tau \left( \frac{2kT_w}{\mu} \right)^{1/2}} \left[ \frac{W}{m^2 K} \right]. \tag{6}
\]

Figure 6 shows temperature contours and streamlines for the $500$ Pa-Tj model with $Q = 500$ W. The streamlines show a recirculation of air that moves upward in the center and downward along the side walls. The asterisk (*) shows the location of the highest speed, which is $0.0125$ cm/sec in the -y direction. The temperature contours show a central hot region with a maximum temperature of $437.2^\circ$C. Although buoyancy induced recirculation is present, the location of the maximum temperature is only slightly above the array center. The $1$ Atm-C and $500$ Pa-C models exhibit similar temperature contours and streamlines as the $500$ Pa-Tj model, but have different maximum speeds and temperatures.

Figure 7 shows the maximum gas speed within the computational domain as a function of array heat load for the three models that include natural convection: $1$ Atm-C, $500$ Pa-C and $500$ Pa-Tj (the Hard Vacuum model does not include convection). Figure 7 shows that the maximum gas speed for all three models increases with heat generation rate. The gas speeds for the $500$ Pa models ($500$ Pa-C and $500$ Pa-Tj) are essentially identical, indicating that the rarified-gas thermal resistance has no significant effect on the gas speed compared to the continuum model. In contrast, the $1$ Atm-C gas speeds are two orders of magnitude higher than the low pressure models due to its higher gas density.

In Fig. 6 a vertical dashed line bisects one of the innermost columns of rods. Figure 8 shows the temperature along that line minus the wall temperature $\Delta T = T - T_w$, versus the y-coordinate. Results are shown for $Q = 500$ W from all four models. For all the profiles, the temperatures within the rods are relatively isothermal compared to the temperature gradients in the gas, and the maximum rod temperatures are near the enclosure mid-height.

The two continuum models ($1$ Atm-C and $500$ Pa-C) exhibit the lowest temperatures. The gas speeds in the $1$
Atm-C model are two orders of magnitude higher than they are at 500 Pa models. The higher gas speed causes the maximum temperature to be located at a slightly higher elevation in the 1 Atm-C calculation than in the 500 Pa-C simulation. However, the profiles are nearly identical. This is because at both pressures the air conductivity is nearly the same and the motion within the gas is so slow that it hardly affects the temperature.

The rarified-gas thermal resistance included in the 500 Pa-T_j model causes the rod and gas temperatures in Fig. 8 to be hotter than they are in the two continuum models. In the Hard Vacuum model, all conduction and convection transport in the gas is eliminated and the rod temperatures are hotter than they are in the 500 Pa-T_j model. At Q = 500 W the maximum temperature difference between the rods and the walls is $\Delta T_{\text{MAX}} = T_{\text{MAX}} - T_{w} = 112.2^\circ\text{C}$ for the 500 Pa-T_j model. For the Hard Vacuum model the maximum temperature difference is 7$^\circ\text{C}$ hotter, and it is 7.5$^\circ\text{C}$ cooler in the 1 Atm-C and 500 Pa-C models.

Figure 9 shows the maximum temperature difference versus array heat generation rate for all four models. While $\Delta T_{\text{MAX}}$ increases with the heat load for all cases, the increase is not linear because the effects of natural convection and radiation heat transfer increase with temperature. As Q increases from 100 to 500 W, the hard vacuum model predicts maximum temperature differences that are larger than the 500 Pa-T_j model by 3 to 7$^\circ\text{C}$, while both continuum models gives maximum temperature differences that are 1.7 to 7.5$^\circ\text{C}$ smaller.

Figure 10 shows the ratio of the temperature difference from the Hard Vacuum, 500 Pa-C, and 1 Atm-C models to that of the 500 Pa-T_j model versus assembly heat generation rate. The continuum models predict maximum temperature difference that are consistently 6% lower than that of the 500 Pa-T_j model, while the hard vacuum model predicts maximum temperature differences that decrease from 11% to 6% higher as the heat load increases.

The experimental apparatus shown in Fig. 1 uses thermocouples to measure enclosure and heater rod temperatures. The uncertainty in the thermocouples measurements is 1.4$^\circ\text{C}$.

As described in an earlier section of this paper, CFD simulations have been performed and compared to experiments conducted using the experimental apparatus shown in Fig. 1 when it was filled with helium and nitrogen gas with pressures between 1 and 3 atm. These simulations predicted maximum rod-to-wall temperature differences that were randomly larger and smaller than the experimental measurements, but are within 7$^\circ\text{C}$ of 95% of the measured values. From the current simulations, at a wall temperature of 325$^\circ\text{C}$, assembly heat generation rate of 500 W, and the apparatus filled with air at P = 500 Pa, the maximum temperature difference predicted by the rarified gas simulations is 7$^\circ\text{C}$ larger than that predicted by the continuum simulations. We wish to use the experimental apparatus to benchmark rarefied gas simulation. In order to do this, the apparatus must be operated under conditions that cause the maximum temperature difference predicted by the rarefied gas model to be more than 7$^\circ\text{C}$ larger than that predicted by the continuum model. Otherwise, it will be extremely difficult to determine if any measured variation from the...
continuum model is caused by rarified gas thermal resistance, or random measurement errors.

One way to increase the difference between the maximum temperatures predicted by the continuum and rarified gas models is to decrease the enclosure temperature. This decreases the fraction of heat transferred from the rods by radiation, increases the fraction carried by gas conduction and convection, and therefore increases the effect of the rarified gas thermal resistance on the maximum temperature difference. Another way is to use larger assembly heat generation rates.

ACKNOWLEDGMENTS

This work was supported by the US Department of Energy Global Nuclear Energy Partnership (GNEP) under contract DEFC07-06ID14782, and the DOE Office of Nuclear Energy’s Nuclear Energy University Program.

REFERENCES
