USE OF REGULAR ROD ARRAYS TO MODEL HEAT TRANSFER FROM BWR FUEL ASSEMBLIES INSIDE TRANSPORT CASKS

Pablo E. Araya  
Research Assistant  
University of Nevada, Reno  
Mechanical Engineering Department  
Mail Stop 312, Reno 89557  
Phone: 775-784-7575  
Email: arayap@unr.nevada.edu

Miles Greiner  
Professor  
University of Nevada, Reno  
Mechanical Engineering Department  
Mail Stop 312, Reno 89557  
Phone: 775-784-4873  
Email: greiner@unr.edu

ABSTRACT  
The current work is a scoping study to determine which heat transfer effects are significant in the fuel/backfill gas region of spent nuclear fuel transport casks. A two-dimensional finite volume mesh that accurately models the geometry of a 7x7 Boiling Water Reactor (BWR) assembly with its channel in a square isothermal enclosure is constructed. The peak cladding temperature is determined using computational fluid dynamics (CFD) simulations for a range of enclosure temperatures, fuel heat generation rates, cladding surface emissivities, and for both nitrogen and helium backfill gases. This work quantifies both the effect of buoyancy induced gas motion in the fuel/backfill gas region and the conditions when it does not significantly affect heat transfer. Future cask design simulations that neglect gas motion will require less computational resources than ones that do not. This work also quantifies the sensitivity of the maximum cladding temperature to fuel cladding emissivity. This helps quantify the uncertainty of temperature predictions if the emissivity is not known. The current CFD technique must be experimentally benchmarked before it may be used with confidence to predict peak cladding temperatures in transport casks. This work indicates that the thermal resistance between a BWR assembly’s channel and the basket walls may be modeled analytically. This will reduce the effort required for benchmark experiments because they will not need to include the channel.

INTRODUCTION  
Spent nuclear fuel is transported in thick walled casks [1,2]. Individual fuel assemblies are supported horizontally within square cross section basket tubes inside the cask containment volume. This volume is evacuated and backfilled with helium or another non-oxidizing gas before shipment. The fuel cladding temperature increases as the fuel heat generation rate increases. Finite element (FE) cask models [3] are used to predict the cladding temperature and determine the maximum heat generation rate that does not cause the cladding to exceed its integrity temperature limit of 400°C [4]. In the past, resources were not available to construct and use computational cask models that accurately represent the fuel geometry. Smeared solids with effective thermal conductivities (ETC) were employed in the fuel regions [1,2,5,6,7,8]. The ETC models conservatively overestimate the cladding temperature. This causes cask FE simulations to underestimate the allowable fuel heat generation rate compared to levels that may be safely transported.

Under-loading casks increases the required number of transport casks and shipments and the associated costs. More accurate (less conservative) predictions of cask capacity can lead to lower system costs. It can also reduce the number of shipments and therefore the risk of transportation accidents. In a recent study, two-dimensional finite volume meshes were constructed that accurately model the geometry of a legal weight truck cask cross-section, including the fuel inside [9]. Low conductivity nitrogen or high conductivity helium gas filled the void spaces within the fuel regions in different models. Steady state simulations were performed for normal, hot day conditions with a range of fuel heat generation rates. Computational fluid dynamics (CFD) simulations were performed that calculate buoyancy driven gas motion, as well as natural convection and radiation heat transfer in the gas filled region. They also calculate conduction heat transfer within the solid components. These results were compared to a finite element model that employed an effective thermal conductivity in the fuel region. The allowable heat generation rate predicted by the CFD model was significantly larger than that predicted by the conservative effective thermal conductivity model.
CFD simulations may provide more accurate results than ETC models. However they must be benchmarked against experimental data before they can be used with confidence to design transport casks.

Experiments have measured the temperature of mock fuel assemblies within isothermal enclosures [10,11,12]. Computational fluid dynamics simulations have been successful at reproducing that data [13,14,15]. Those experiments and simulations have helped develop an understanding of combined natural convection and radiation heat transfer within the fuel assemble/backfill gas regions of casks. However, they were performed with enclosure temperatures that are significantly lower than ones relevant to casks with fuel cladding temperatures close to their limit value [3,9]. It is known that the relative importance of natural convection heat transfer decreases compared to radiation as the temperature increases. However, the conditions under which natural convection is negligible in the enclosed array geometry have not been determined.

A factor that makes experimentally benchmarking the simulations results difficult is the wide variety of boiling water reactor (BWR) and pressurized water reactor (PWR) fuel assembly geometries that must be considered. In general, these assemblies consist of fuel rods held in square arrays by periodic spacer plates [16]. The rods themselves are stacks of UO$_2$ fuel pellets within zircaloy cladding. Different BWR assemblies consist of 6×6 to 11x11 arrays, and PWR assemblies generally consist of 14x14 to 17x17 rod arrays [16]. PWR rod diameters are generally smaller those of BWRs, but their total assembly cross sections are larger.

While assemblies are for the most part parallel arrays of heating rods, there are some inhomogeneities. For example some spaces within the array contain hollow instrumentation or guide thimble tubes instead of fuel rods [16]. Moreover, BWR rod arrays are surrounded by a square-cross-section zircaloy channel [16]. It is not currently known how these inhomogeneities affect the heat transfer.

**Current Work**

In the current work, heat transfer from a General Electric (GE) 7x7 BWR fuel assembly to an isothermal basket tube is studied numerically. The cross section of this assembly within a standard basket tube is shown in Figure 1 [6,16]. The assembly consists of 49 fuel rods in a square array with 18.75 mm center-to-center spacing. Each rod consists of 12.37 mm diameter uranium dioxide (UO$_2$) fuel pellets within zircaloy cladding. The cladding outer diameter and wall thickness are 14.3 mm and 0.81 mm, respectively. There is an average gap of 0.15 mm between the pellets and cladding. The rod array is surrounded by a 2.03 mm thick, zircaloy channel. The channel outer dimension is 134.1 mm and it has rounded corners with inner radii of 9.7 mm. The entire assembly is placed within a 152.4 mm (6 inches) square basket tube constructed from stainless steel [6]. In this work the fuel pellets are centered within the cladding and the assembly is centered within the basket.

Heat is transferred from the fuel rods to the basket by natural convection and radiation heat transfer across the gas filled regions, and conduction heat transfer through the assembly spacer plates. In this work transport through the spacer plates is not included.

**Figure 1** Schematic of the GE 7x7 BWR fuel assembly within a transport cask basket.

Maximum cladding temperatures are determined for both helium (He) and nitrogen (N$_2$) backfill gases as functions of fuel assembly heat generation rate, basket wall temperature, and cladding emissivity. Computational fluid dynamics (CFD) simulations calculate the buoyancy induced gas motion as well as the natural convection and radiation heat transfer. Stagnant-CFD simulations are also performed to calculate transport with the gas velocity set to zero. Comparing the CFD and stagnant-CFD results shows the contribution of gas motion.

Two different models of the domain in Fig. 1 are developed. In the first, CFD and stagnant-CFD simulations are performed in the entire domain. This is known as the **Full** model. The second is a **Two-Region** model. One of these regions is the **Exterior**, which is between the channel inner wall and inner surface of the basket tube. Conduction and radiation transport in that region is modeled analytically. The second region is the **Interior**, which is inside the channel inner wall. That region is modeled using both CFD and stagnant-CFD simulations.

The results of the Full and Two-Region models are compared to determine the accuracy of the two-region model. If the two-region model is sufficiently accurate, then the relatively expensive computational fluid dynamics and experimental resources of future studies can be focused on the regular rod array of the interior region, and will not need to be used on the exterior. This can be useful because a wide variety of fuel assembly designs must be considered. Reducing the size of the domain where expensive resources must be expended will make that effort easier.

**NOMENCLATURE**

\[ \Delta T_{C,\text{MAX}} = T_{C,\text{MAX}} - T_{c,\text{b}}; \text{ Maximum cladding to enclosure wall temperature difference} \ [\degree{C}] \]
\[
\Delta T_{\text{EXT}} = T_{\text{C,I}} - T_{\text{B}}; \text{ Temperature difference between average internal surface of the channel and basket wall [°C]}
\]
\[
\Delta T_{\text{INT}} = T_{\text{MAX}} - T_{\text{C,I}}; \text{ Temperature difference from maximum to internal surface of the channel [°C]}
\]
\[
\Delta T_{\text{MAX}} = T_{\text{MAX}} - T_{\text{B}}; \text{ Maximum rod to enclosure wall temperature difference [°C]}
\]
\[
\varepsilon \quad \text{Surface emissivity [-]}
\]
\[
k_{\text{CH}} \quad \text{thermal conductivity of channel [W/mK]}
\]
\[
P \quad \text{Peaking Factor [-]}
\]
\[
Q \quad \text{Total assembly heat load [W]}
\]
\[
Q_{\text{CH}} \quad \text{Heat conducted through channel [W]}
\]
\[
Q_{\text{COND}} \quad \text{Heat conducted through gas between channel and basket [W]}
\]
\[
Q_{\text{RAD}} \quad \text{Heat radiated between channel and basket [W]}
\]
\[
\sigma \quad \text{Stefan-Boltzmann constant } 5.67 \times 10^{-8} \text{ [J/K}^4 \text{m}^2\text{s]}
\]
\[
S_{\text{CH}} \quad \text{Conduction Shape factor of channel [m]}
\]
\[
S_{\text{GAS}} \quad \text{Conduction Shape factor of gas between channel and basket [m]}
\]
\[
T_{\text{B}} \quad \text{Basket outer wall temperature [°C]}
\]
\[
T_{\text{MAX}} \quad \text{Maximum rod temperature [°C]}
\]
\[
T_{\text{C,MAX}} \quad \text{Maximum rod cladding temperature [°C]}
\]
\[
T_{\text{C,I}} \quad \text{Inner channel surface temperature [K]}
\]
\[
T_{\text{C,O}} \quad \text{Outer channel surface temperature [K]}
\]
\[
V \quad \text{Volume of UO}_2 \text{ in the Assembly [m}^3\text{]}
\]

**COMPUTATIONAL METHODS**

**Full Model**

Figure 2a shows a detail of the bottom right corner of the two-dimensional finite volume mesh of the Full model. It was constructed using GAMBIT, a part of the FLUENT computational fluid dynamics package. Figure 2a shows a portion of the fine grid, which consists of 30,080 elements. Nominal and coarse grids (not shown) with 15,432 and 7,520 elements, respectively, were constructed to examine mesh sensitivity.

Heat is generated uniformly within the fuel pellets at a volumetric rate of \( q = Q \times P/V \). In this expression \( Q \) is the total assembly heat generation, and the total volume of the UO\(_2\) fuel pellets is \( V = 0.0281 \text{ m}^3 \). The peaking factor \( P = 1.25 \) [6] is used to account for the higher heat generation at the axial center of a BWR assembly compared to the average heat generation rate. A uniform basket wall temperature, \( T_{\text{B}} \), is applied to the enclosure surface.

Temperature dependent thermal conductivities are used for uranium dioxide, zircaloy, helium and nitrogen. A contact resistance is used to model the gap between the fuel pellets and cladding [14]. The emissivity of the stainless steel basket surface is \( \varepsilon_B = 0.6 \). The baseline emissivity for zircaloy is \( \varepsilon = 0.806 \) [6]. Simulations are also performed for a range of cladding emissivities between \( \varepsilon = 0.6 \) to 0.9 [17] to evaluate the sensitivity of temperature results to this property.

**Two-Region Model**

In the two-region model, the computational domain is divided into interior and exterior of the inner channel surface. The channel interior and exterior surface temperatures, \( T_{\text{C,I}} \) and \( T_{\text{C,O}} \), respectively, are modeled as isothermal.

\[
\text{The interior region is modeled analytically. The rate of conduction heat transfer through the channel is calculated as}
\]
\[
Q_{\text{CH}} = S_{\text{CH}} k_{\text{CH}} (T_{\text{C,I}} - T_{\text{C,O}})
\]

(1)

In this expression the channel conduction shape factor \( S_{\text{CH}} = 948 \text{ m} \) was determined from a numerical simulation [18], and the conductivity of the zircaloy channel is \( k_{\text{CH}} = 12.8 \text{ W/mK} \) at 25°C.

The heat transferred through the channel, \( Q_{\text{CH}} \), is equal to the combined conduction and radiation heat transfer across the gas filled void between the outer channel surface and the basket surface.

\[
Q_{\text{CH}} = Q_{\text{COND}} + Q_{\text{RAD}}
\]

(2)

The conduction and radiation components are calculated as

\[
Q_{\text{COND}} = S_{\text{GAP}} k_{\text{GAS}} (T_{\text{C,O}} - T_{\text{B}})
\]

(3)

\[
Q_{\text{RAD}} = \frac{\sigma}{4} \left( T_{\text{C,O}}^4 - T_{\text{B}}^4 \right)
\]

(4)

In the conduction expression the gap conduction shape factor \( S_{\text{GAP}} = 277 \text{ m} \) was determined from a numerical simulation [18], and the thermal conductivity of the backfill gas \( k_{\text{GAS}} \) is dependent on the average channel wall temperature. In the radiation expression, the basket surface emissivity is \( \varepsilon_B = 0.6 \), \( A_{\text{C,O}} = 1.95 \text{ m}^2 \) is the channel outer surface area and \( A_{\text{B}} = 2.23 \text{ m}^2 \) is the basket surface area. Equations (1)-(4) are used to determine the inner channel surface temperature for different total heat loads \( Q_{\text{CH}} \) and basket surface temperatures, \( T_{\text{B}} \).

Figure 2 (a) shows a detail of the fine mesh for the Full model and (b) detail of the fine mesh for the Square Cornered Isothermal (SCI) model.

The interior region is modeled as a regular rod array within a square-cornered isothermal (SCI) enclosure. CFD and stagnant-CFD simulations are performed for the computational
domain whose lower right corner is shown in Fig. 2b. In that model, the rounded corners of the channel have been squared off, and the non-uniform channel temperature is modeled as isothermal. This is closer to the type of boundary conditions that are practical for experimental benchmarking purposes than the actual channel conditions. Figure 2b shows a fine meshed SCI model consisting of 26944 elements.

RESULTS AND DISCUSSION

Full Model

Figure 3 shows temperature contours within the full domain for a heat generation rate of \( Q = 600 \) W with \( \text{N}_2 \) backfill. Figures 3a and 3b show results from the stagnant-CFD and CFD simulations, respectively. The right side of Fig. 3b shows streamlines of the flow field.

For the stagnant gas simulations in Fig. 3a, the hottest location is at the center of the domain. The hottest cladding temperature \( T_{\text{C,MAX}} \) exists at four locations; the top, bottom and both sides of the center fuel rod. The right side of Fig. 3b shows that when buoyancy-induced motion is included, currents move upward in the center of the rod-filled portion of the domain and downward at its edge. Slower gas motion with some recirculation also exists in the gap between the channel and basket. The left side of Fig. 3b shows that this gas motion causes the hottest location to be above the domain center.

Figure 4 shows the maximum flow speed, \( S \), in the domain for CFD simulations with \( \text{N}_2 \) and He as function of assembly heat generation rate. Results from the Full and SCI domains are presented. For the Full model at high basket wall temperatures, the flow speeds due to natural convection are low \((< 2 \text{ cm/s})\), whereas for \( \text{N}_2 \) at \( T_B = 25^\circ\text{C} \), the flow speeds reach up to 13.7 cm/s for \( Q = 1000 \) W.

Figure 5 is a plot of the temperature along the domain vertical centerline (the line connecting points A and B in Fig. 1) versus distance from the bottom of the domain, \( Y \). Results from the CFD (solid lines) and stagnant-CFD (dashed lines) simulations are plotted for nitrogen with heat loads of \( Q = 100\)W, 500W and 1000W.

Each profile exhibits seven large flat regions in the domain interior, which represent the temperature within fuel rods. There are also two smaller flat regions near the domain edge that are in the channel. The nitrogen gas between these solid components exhibits larger temperature gradients due to its relatively low thermal conductivity.

The stagnant-CFD profiles are symmetrical with respect to the domain center. Buoyancy driven gas motion in the CFD simulations makes the top two rows hotter than the stagnant-CFD results. However, for each heat generation rate, gas motion reduces the maximum cladding temperature, \( T_{\text{C,MAX}} \), compared with the stagnant-gas simulations. For \( Q = 500 \) W, the maximum cladding to basket wall temperature difference is \( \Delta T_{\text{C,MAX}} = T_{\text{C,MAX}} - T_B = 100^\circ\text{C} \) for the stagnant gas simulations. Buoyancy induced gas motion reduces it to 95.3°C.

Figure 6a shows \( \Delta T_{\text{C,MAX}} \) versus \( Q \) from the CFD and stagnant-CFD simulations for both helium and nitrogen. Results from three mesh refinements are shown for the CFD simulations. The small difference between these results verifies mesh independence.
For both the CFD and stagnant-CFD simulations $\Delta T_{c,max}$ increases with Q. However, the slope decreases as Q increases because the relative contribution from radiation and natural convection heat transfer increases at higher temperatures. For each backfill gas, $\Delta T_{c,max}$ is smaller for $T_B = 400\,^\circ C$ than it is for $T_B = 25\,^\circ C$. This is once again due to the increased importance of radiation heat transfer at higher temperatures. Finally, for a given wall temperature, $\Delta T_{c,max}$ is smaller for helium than it is for nitrogen. This is caused by helium’s larger thermal conductivity.

Figure 6b shows the ratio of the maximum cladding to wall temperature difference from the CFD simulation to that from the stagnant-CFD, $R = \Delta T_{c,max,CFD}/\Delta T_{c,max,STAG}$. Helium results are presented for $T_B = 25\,^\circ C$ and $400\,^\circ C$, while nitrogen results are presented for $T_B = 25\,^\circ C$, $100\,^\circ C$ and $400\,^\circ C$. The ratio is less than unity when the buoyancy induced gas motion increases the total heat transfer relative to a stagnant gas.

For all helium and the higher temperature nitrogen simulations the ratio is near unity. This indicates that the high conduction and radiation transport associated with these conditions are only mildly enhanced by buoyant effects. However, for Nitrogen with $T_B = 25\,^\circ C$, buoyancy effects reduce the maximum cladding to wall temperature difference by 5.6% at Q = 300 W. For $T_B = 100\,^\circ C$, the maximum reduction in the ratio is 1.7% at Q = 600 W.

The ratio approaches unity for both high and low values of the assembly heat generation rate. At low generation rates buoyancy effects are not sufficiently strong to increase heat transfer relative to the levels from conduction. At higher heating loads, the effects of radiation begin to dominate over those of either natural convection or conduction. This causes the temperatures from the CFD and stagnant-CFD calculations to be nearly the same.

These results suggest that if helium is the backfill gas and/or the basket wall temperature is greater than $100\,^\circ C$, buoyancy induced gas motion does not significantly affect heat transfer. However, at lower temperatures, the effects of buoyancy can be significant.
transfer, and stagnant-CFD simulations are sufficient to characterize the heat transfer in spent nuclear fuel assemblies.

Figure 7 is a plot of the temperature difference between the inner channel surface and the basket versus channel surface coordinate, s. Figure 1 shows the origin of the s-coordinate at the center of the top channel surface. Results are for Q = 100 and 1000 W/assembly with nitrogen and T_B = 25°C. Vertical dotted lines show the corners between the top, side and bottom portions of the channel. The temperature along only one-half of the channel is plotted due to symmetry.

For the stagnant-CFD results, the hottest channel locations are at the center of the top, bottom and side surfaces, and the coolest are at the channel corners. For the CFD simulations, buoyancy induced gas motion causes the channel temperature to be hotter on the upper surface and cooler on the bottom than the stagnant gas simulations. The differences at the top and bottom are nearly equal for Q = 100 W. The difference is significantly larger on the bottom than on the top for Q = 1000 W. As Q increases, the channel inner-surface average temperature, $T_{CH,1}$, and its non-uniformity (difference between its maximum and minimum values) both increase.

Two Region Model

We wish to determine if the full domain in Fig. 1 can be effectively divided into an external and internal region. To do this we divide the maximum temperature difference $\Delta T_{MAX} = T_{MAX} - T_B$ into two components $\Delta T_{EXT} + \Delta T_{INT}$. The first is the external temperature difference, which is the difference between the average channel inner surface temperature and the basket temperature, $\Delta T_{EXT} = \overline{T_{CH,1}} - T_B$. The second is the internal temperature difference between the maximum fuel temperature and the average temperature of the inner channel surface, $\Delta T_{INT} = T_{MAX} - \overline{T_{CH,1}}$. The triangular symbols in Figs. 8 and 9 show, respectively, $\Delta T_{EXT}$ and $\Delta T_{INT}$ versus Q for different wall temperatures and gases. The solid triangles are from CFD simulations while the open triangles are from stagnant CFD simulations. While buoyancy induced fluid motion reduces these temperature differences relative to the stagnant gas results, the reduction is only significant for nitrogen with $T_B = 25°C$.

The results for $\Delta T_{EXT}$ and $\Delta T_{INT}$ exhibit similar trends as the maximum cladding temperature difference $\Delta T_{C,MAX}$ presented in Fig. 6a. Comparison of Figs. 6a and 8 shows that the exterior temperature difference (between the channel interior and the basket) is 28% to 43% as large as the maximum temperature difference (between the hottest cladding location and the basket). The region between the inner channel surface and basket surface represents a significant portion of the total thermal resistance between the hottest fuel rod and the basket.

The lines in Fig. 8 show the channel to basket wall temperature difference from the analytical model from Equations (1) to (4). This model is within 4.6% of the CFD results from the full model. The circles in Fig. 9 show the internal temperature difference calculated in the SCI (square corner isothermal) domain described in Fig. 2b. The solid circles are from the CFD simulations while the open circles are from the stagnant-CFD calculations. The boundary temperature for each simulation was determined from the analytical external model results presented in Fig. 8. The CFD and stagnant-CFD results
from the SCI domain are within 1.1% of the corresponding results from the full domain.

Fig. 4 also shows maximum speeds for the SCI model. In all cases the maximum speed is slightly higher in the SCI model that in the Full model.

The circles in Figure 10 show the maximum fuel temperature calculated from the two-region model, \( \Delta T_{\text{MAX}} = \Delta T_{\text{EXT}} + \Delta T_{\text{INT}} \) versus \( Q \) for both wall temperatures and both gases. The solid symbols are from CFD simulations while open circles are for the stagnant-CFD results. Solid and open triangles show the corresponding results from the full model from Fig. 6a. The two-region model results are within 1.4% of the full model CFD results, and within 1.4% of the stagnant-CFD results. The close agreement between the results of these two modeling methods indicates that the non-homogeneity does not need to be included in benchmark experiments.

Figure 11 shows the sensitivity of the internal temperature difference to different cladding emissivities. It is a plot of the internal temperature difference divided by the temperature difference with clad emissivity of \( \varepsilon_c = 0.806 \) (the baseline value used in this work) versus \( \varepsilon \). Figures 11a and 11b show results for nitrogen and helium, respectively. For each plot results are presented for \( T_B = 25^\circ \text{C} \) and \( 400^\circ \text{C} \). Data for different heat generation rates are nearly identical. Results from the SCI and full domain models are included. They are within 1.6% of each other.

For a basket temperature of \( T_B = 400^\circ \text{C} \), a 10% increase in \( \varepsilon_c \) causes a 7.2% decrease in \( \Delta T_{\text{INT}} \) for nitrogen, and a 5.3% decrease for helium. The nitrogen results are more sensitive to variations in cladding emissivity than helium. For nitrogen, a large fraction of the heat is transferred by radiation, compared to that in helium, because of nitrogen’s lower thermal conductivity. For both gases, the cladding temperature is less sensitive to it’s emissivity at the lower wall temperature due to the decreased importance of radiation heat transfer.

**SUMMARY AND CONCLUSIONS**

Highly radioactive spent nuclear fuel is transported in thick walled casks. It is placed in square cross-section tubes of a basket structure inside the cask containment volume. This volume is evacuated and backfilled with a non-oxidizing gas. Heat generated by the fuel is transferred through the gas and cask walls to the environment. The walls must be thick enough to protect the public, but must be conductive enough so that the fuel cladding does not exceed its allowed temperature. Finite element (FE) thermal models are used to determine the cask thermal capacity, which is the maximum fuel heat generation rate that does not cause the fuel cladding to exceed its temperature limit.

Effective thermal conductivity (ETC) models are typically employed in the fuel regions of the FE models to predict the cladding temperature. They model conduction and radiation heat transfer from the fuel rods to the basket but neglect some heat transfer effects. It is difficult to develop conductivities that are accurate for different fuel assembly designs, backfill gas compositions, pressures, and fuel cladding emissivities. Conservative assumptions are therefore employed so that the conductivities overestimate the cladding temperature and FE models that employ the conductivities underestimate the cask capacity.

Computational fluid dynamics (CFD) simulations offer an opportunity to more completely model fuel/backfill gas regions than ETC’s, but they require more computational resources. The current work is a scoping study to determine which heat transfer effects are significant and which may be neglected. A two-dimensional finite volume mesh that accurately models the geometry of a 7x7 Boiling Water Reactor (BWR) assembly with a surrounding channel in a square isothermal enclosure is constructed. The peak cladding temperature is determined for a
range of basket temperatures, fuel heat generation rates, cladding surface emissivities, for both nitrogen and helium backfill gas. This technique must be experimentally benchmarked before it may be used with confidence to predict peak cladding temperatures in transport casks.

This work quantifies both the effect of buoyancy induced gas motion in the fuel/backfill gas region and the conditions when it does not significantly affect heat transfer. Future cask design simulations that neglect gas motion will require less computational resources than ones that do not. This work also quantifies the sensitivity of the maximum cladding temperature to fuel cladding emissivity. This helps quantify the uncertainty of temperature predictions if the emissivity is not known. Finally, this work indicates that the thermal resistance between a BWR assembly’s channel and the basket walls may be modeled analytically. This will reduce the effort required for future benchmark experiments because they will not need to include the channel.

ACKNOWLEDGMENTS

This work was sponsored by the US Department of Energy through Sandia National Laboratories under contract DE-C04-94AL85000.

REFERENCES


