CFD SIMULATIONS OF NATURAL CONVECTION/RADIATION HEAT TRANSFER WITHIN THE FUEL REGIONS OF A TRUCK CASK FOR NORMAL TRANSPORT

Venkata V.R. Venigalla
Research Assistant
Mechanical Engineering Department
University of Nevada Reno
Reno, Nevada 89557
venigall@unr.nevada.edu

Miles Greiner
Professor of Mechanical Engineering
University of Nevada, Reno
Reno, NV 89557
Phone-(775) 784-4873
greiner@unr.edu

ABSTRACT
A two-dimensional finite volume mesh is constructed that accurately represents the geometry of a legal weight truck cask, including four PWR fuel assemblies inside. CFD simulations calculate buoyancy driven gas motion as well as natural convection and radiation heat transfer in the gas filled fuel regions. They also calculate conduction within the cask solid components. The cask is in a normal transportation environment. The fuel and cask temperatures are calculated for ranges of fuel heat generation rate and cladding emissivity, for both helium and nitrogen backfill gas. The cask thermal capacity, which is the fuel heat generation rate that brings the peak fuel cladding to its temperature limit, is also determined. The results are compared to simulations in which the gas speed is set to zero, to determine the effect of buoyancy induced motion. The allowable heat generation rate is 23% higher for helium than for nitrogen due to helium’s higher thermal conductivity. Increasing the cladding emissivity by 10% increases the allowed fuel heating rate by 4% for nitrogen, but only 2% for helium. The higher value for nitrogen is caused by the larger fraction of heat transported by radiation when it is the backfill gas compared to helium. The stagnant-gas calculations give only slightly higher cladding temperatures than the gas-motion simulations. This is because buoyancy induced gas motion does not greatly enhance heat transfer compared to conduction and radiation for this configuration. The cask thermal capacity from the stagnant-CFD calculation is therefore essentially the same as that from the CFD simulation. This suggests that future cask thermal calculations may not need to include gas motion. These results must be experimentally benchmarked before the CFD methods can be used with confidence for designing transport casks. Basket surface temperatures calculated in this work can be used as the basis for boundary condition in those experiments.

INTRODUCTION

Light water reactor nuclear fuel assemblies consist of fuel rods held in square arrays by periodic spacer plates [1, 2]. The rods themselves are stacks of UO2 fuel pellets within zircaloy cladding. Some spaces in the array contain hollow instrumentation or guide thimble tubes instead of fuel rods. Different boiling water reactor (BWR) assemblies consist of 6×6 to 9×9 arrays of rods surrounded by a square-cross-section zircaloy channel. Pressurized water reactor (PWR) assemblies generally consist of 9×9 to 17×17 rod arrays but do not have surrounding channels. PWR rod diameters are generally smaller than those of BWRs, but their total assembly cross sections are larger.

Spent nuclear fuel (SNF) is placed in water pools after it is removed from a reactor to allow its heat generation and radioactive decay rates to decrease [3]. After an appropriate time SNF is placed in casks for dry storage or offsite transport [4, 5]. In transportation casks, individual SNF assemblies are supported horizontally within square cross-section basket tubes inside the cask’s containment region. That region is evacuated and backfilled with helium or another non-oxidizing gas. Casks transported by truck or rail have enough space for roughly 4 or 21 PWR assemblies, respectively [4, 5].

Heat generated by the SNF assemblies makes the cask and fuel hotter than their surroundings [6, 7]. However, the zircaloy cladding that encapsulates the fuel pellets provides an important boundary and its temperature must not exceed its containment integrity limit of 400°C [8]. This can limit the cask thermal capacity, which is the number and heat generation rate of assemblies that may be safely loaded.

Finite element thermal simulations are used to predict cask and fuel temperatures for a given fuel heat generation rate [4]. Multiple simulations are used to determine the maximum fuel heat generation rate that does not cause the fuel cladding to exceed its temperature limit [6]. The multiple fuel regions of a cask are complex to model because each contains many fuel rods. In the past computational resources were not available to perform multiple calculations in finite element models that accurately represented the fuel. To address this problem, the fuel assemblies and backfill gas were replaced by fictitious but representative solid elements with temperature-dependent effective thermal conductivities (ETC) [4, 9-11].

Manteufel and Todreas [9] developed ETC models based on a one-dimensional analytical model of radiation and conduction heat transfer in a two-dimensional array of heated rod within a stagnant gas. That model does not account for unheated tubes or channels, or multi-dimensional effects. Bahney and Lotz [10] performed two-dimensional finite element thermal simulations of
several different fuel assemblies, including unheated components, within isothermal enclosures. They developed ETC models based on conduction and radiation heat transfer. In some cases ETC models are employed in analysis without stating their source or the methods used to develop them [4, 11].

A shortcoming of conductivity models is that they approximate heat flux at a location based only on the temperature and its spatial gradient at that location. This is not universally appropriate. Natural convection heat flux is dependent on local velocity, which depends on temperatures at different locations. Radiation heat transfer also depends on temperatures at a distance. As a result, a conductivity model that is appropriate within fuel basket openings near the center of a large rail cask, where the surface is nearly isothermal, may not be accurate for tubes near its periphery. Simulations have shown that the basket temperature near the periphery of rail cask is highly non-uniform [7]. This is important because the thermal resistance of the periphery cells strongly affect the temperature of the center basket opening, which affects the hottest cladding.

The simplifying assumptions employed in ETC models are intended to cause them to over-predict the cladding temperature. Finite element cask models that employ this methodology therefore under-predict the cask thermal capacity. This can cause operators to under-load cask compared to their true safe capacity. This increases the required number of storage and transport casks, the number of 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.

Experiments that measure the temperature within mock fuel assemblies inside isothermal enclosures have provided a good understanding of the combined natural convection and radiation heat transfer within the fuel assemble/backfill gas regions of casks [12-14]. Computational fluid dynamics (CFD) simulations of those experiments accurately reproduced that data [15-18] and hold promise for developing improved fuel region heat transfer models.

Currently available computational resources are sufficient to perform two-dimensional simulations in a full cask cross section that employ CFD in the multiple fuel regions. This can lead to better estimates of the cask thermal capacity for specific fuel and cask designs, even for non-uniform fuel heat generation. It also provides an opportunity to accurately determine the non-isothermal temperature profiles of the periphery basket tubes [6].

In the current work two-dimensional models of a legal weight truck (LWT) cask cross-section are developed. Four PWR assemblies, each consisting of a 15x15 square array of heat generating fuel rods, are placed within the basket tubes. Buoyancy induced gas motion and Natural convective/radiation heat transfer within the fuel regions, as well as conjugate conduction in the solid regions, are modeled using the Fluent CFD package. The results are compared to those from finite element models that employ ETC models in the fuel regions.

The maximum allowed heat generation rate that causes the cladding to reach its temperature limit is determined for atmospheric pressure helium and nitrogen backfill gases. The sensitivity of the allowable heat generation rate to the cladding emissivity is also evaluated. The non-uniform temperature profiles of the basket tubes are determined at the maximum allowed heat generation rate. These profiles will be used in future benchmark experiments.

**NOMENCLATURE**

- BWR: Boiling water reactor
- CFD: Computational Fluid Dynamics
- ETC: Effective thermal conductivity
- LWT: Legal Weight Truck
- PWR: Pressurized Water Reactor
- SNF: Spent Nuclear Fuel
- Q: Heat generation rate within an individual fuel assembly
- Q_A: Maximum allowable heat generation rate at which the cladding reaches the temperature of concern
- S_max: Maximum flow speed
- T_avg: Temperature along the walls of fuel basket
- ΔT_{NU}: Average of the temperature between the center and 14.5 mm of fuel basket
- T_{CL}: Temperature of cladding concern
- ε_{c}: Cladding surface emissivity
- ε_{SS}: Support Structure surface emissivity
- ε_{SS}:

**LEGAL WEIGHT TRUCK PACKAGE**

Figure 1 shows the cross section of a generic Legal Weight Truck (LWT) cask loaded with four spent PWR fuel assemblies. This cask is similar but not identical to a currently licensed package [4]. The model is developed using MSC/Patran. The cross section in Fig. 1 is midway between cask ends. The dot-filled region represents four 15x15 PWR fuel assemblies within backfill gas. All 225 fuel rods of each assembly are identical. Each contains UO_2 pellets of diameter 9.36 mm enclosed in zircaloy cladding of thickness 0.78 mm. No gap or contact resistance between the pellets and cladding are modeled. The square tube array has center to center spacing of 14.5 mm, and the distance between the center of the outer most rod and the basket wall is 9.82 mm. The fuel assembly is similar to a Babcock & Wilcox 15x15 Mark B PWR [19], but the model does not contain unheated components.

The cross-shaped component at the center of the package is a 1.5 cm thick stainless steel support structure. Its surface emissivity is ε_{SS} = 0.8. Borated carbon (B,C) pellets fill 1.1 cm-diameter holes that are drilled radially in the legs of the structure. The sides of the four square openings where the fuel is placed are H = 22.3 cm long.

The support structure and fuel are surrounded by a 0.96 cm thick stainless steel liner. Its emissivity is ε_{SS} = 0.2. The liner is surrounded by a depleted uranium gamma shield. Its maximum thickness is 6.7 cm and it has an outer radius of curvature of 11.4 cm at its corners. A 3.8 cm thick stainless steel package body surrounds the gamma shield.

An external, 12.4-cm-thick neutron shield encircles the package. A single region is used to model several components. These components are 12.1-cm-thick Polypropylene-1% boron, 24 aluminum radial fins of thickness 0.245 cm, and a 0.27 cm thick stainless steel outer skin. The mixture thermal conductivity for this composite structure was developed based on an equivalent conduction model [20].

Heat generated within the fuel rods is transferred to the surrounding surfaces by natural convection and radiation heat transfer across the backfill gas, and conduction through the rod spacer plates. In this work we neglect the spacer plates.
COMPUTATIONAL METHODS

Cask Models

Figure 2a shows the computational meshes used in the current work. Only one half of the cask cross section is modeled to take advantage of the geometric and boundary condition symmetry. The computational grid is constructed using MSC Patran software. Mesh independence is evaluated using coarse and fine grids with 46,798 and 210,874 elements, respectively. Portions of the fuel region of those meshes are shown in Figs. 2b and 2c. The majority of simulations documented in this work are performed using the coarse grid.

The current simulations calculate the cask and fuel temperatures using three different fuel region models. One model uses computational fluid dynamics (CFD) simulations that include buoyancy-induced fluid motion. Another uses CFD but assumes the gas speed is zero (Stagnant-CFD). Both these models include the effects of radiation heat transfer. Comparison of these two results shows the effect of gas motion. The last model employs the effective thermal conductivity (ETC) developed by Manteufel and Todreas [9].

Different simulations are performed for each model using helium or nitrogen backfill gases. Temperature dependent thermal conductivities are applied to all solid and gas components. Those properties were obtained from standard sources [4, 5]. The support structure and stainless steel liner emissivities are $\varepsilon_{SS} = 0.8$ and $\varepsilon_{SS} = 0.2$, respectively. The cladding emissivity used in the majority of simulations is $\varepsilon_c = 0.8$. Several different cladding emissivities are presented in the literature [9, 11]. Simulations were performed for the range $\varepsilon_c = 0.6$ to 0.9 to determine its effect on the results.

For both the CFD and Stagnant-CFD models the meshes in Fig. 2 are imported into the Fluent CFD package. That code utilizes the finite volume method to solve the governing mass, momentum and energy equations. A second order upwind discretization scheme is used to solve the momentum and energy equations along with SIMPLEC algorithm for pressure velocity coupling.

For the CFD simulations, Fluent calculates buoyancy induced gas motion, convection and surface-to-surface radiation heat transfer across the gas filled regions, and conduction in the solid components. For the Stagnant-CFD simulations the gas speed is set to zero. Those results were compared to determine the effect of gas motion.

For all Fluent calculations a uniform volumetric heat generation was applied to all the fuel pellets. It was determined by dividing the total heat generation rate of each assembly by the total volume of its fuel pellets (0.05 m$^3$, based on a rod length of 3.601 m [19]), multiplied by a peaking factor of 1.25 [6, 10]. That factor accounts for the higher heat generation rate in the midsection of the PWR assembly compared to its average.

The ETC simulations are performed using the Patran P/thermal finite element code. The Manteufel and Todreas model [9] is applied to the 15x15 PWR assembly used in the current work. In that model the region inside each square opening of the fuel basket is divided into a central square that represents the fuel assembly, and a thin edge region that represents the gas filled gap between the assembly and basket. For the current work the outer edge thickness is 0.3 cm, which is the distance between the outermost rod center and the wall, minus half the pin center-to-center pitch. The interior square with side length 21.7 cm represents the fuel.

Figure 1: Cross section of a Legal Weight Truck (LWT) cask that transports four spent pressurized water reactor fuel assemblies.

Figure 2: Computational domain of a LWT cask (a) Half Coarse Grid (b) Detail of fine grid with 210874 elements (c) Detail of Coarse grid with 46798 elements.

Figure 3: Effective thermal conductivity versus Temperature for Helium and Nitrogen backfill gases.
The Manteufel and Todreas model gives two different effective thermal conductivities for the interior and edge regions $k_{\text{INT}}$ and $k_{\text{EDGE}}$. These conductivities are plotted versus temperature in Fig. 3 for both helium and nitrogen backfill gases. The interior and edge conductivities were applied to the appropriate elements in Fig. 2.

A uniform volumetric heat generation rate is applied to the central square region. It is determined by dividing the total assembly heat generation rate by the effective fuel volume (area of the square region times its effective length), multiplied by a peaking factor.

**Cask Boundary Conditions**

For all simulations the cask exterior surfaces are exposed to normal hot day conditions [21]. Those conditions are still air at an environment temperature of $T_e = 38 ^\circ C$, with a constant, solar heat flux of 388 W/m² absorbed by the entire exterior package surface. In this work, the solar absorptivity is unity and the package radiates to the environment with a surface emissivity of 0.2 [4]. The natural convection heat flux at each surface location is calculated as $q = h (T - T_e)$. In this expression $T$ is the local surface temperature, and $h$ is the heat transfer coefficient between the package surface and its surroundings. It is determined based on a Nusselt number correlation for a horizontal cylinder in stagnant air [22]:

$$ h = \frac{k}{\delta} \left( 0.6 + \frac{0.387 Ra^{1/6}}{1 + \left( \frac{0.559}{Pr} \right)^{29/16}} \right)^2 $$

(1)

In this equation the package diameter is $d = 2.42$ m, $k$ is the thermal conductivity of air, $Pr = \nu/\alpha$ is the air Prandtl number. The kinematic viscosity of air is $\nu$ and its thermal diffusivity is $\alpha$. The Rayleigh number is

$$ Ra_d = \frac{g \beta (T - T_e) \delta^3}{\nu \alpha} $$

(2)

where $g$ is the acceleration of gravity and $\beta$ is the isobaric expansion coefficient of air.

**SIMULATION RESULTS**

**Q = 800 W**

**Fuel and Cask Temperatures** Figure 4 shows temperature contours within the right side of the cask for a fuel heat generation rate of Q = 800 watts/assembly. The plots in Figs. 4a, 4b and 4c are from the ETC, Stagnant-CFD and CFD simulations, respectively. The temperatures in the left side of the cask are a mirror image of those in the figure and are not shown.

Buoyancy-induced gas motion is not included in the ETC or Stagnant-CFD simulations. As a result the temperature contours exhibit the same symmetry as the cask geometry. The hottest fuel is located on diagonal lines that are 45° above and below horizontal, and the maximum temperatures in the upper and lower fuel regions are the same. The walls that enclose the fuel are not isothermal. The sides that are closer to the package outer surface are cooler than those closer to the interior.

The ETC temperature contours in Fig. 4a exhibit discontinuities at the lines between the interiors and edges of the fuel regions. This is caused by the difference in effective conductivity in those regions. The Stagnant-CFD and CFD temperature contours in Figs. 4b and 4c exhibit discontinuities throughout their fuel regions. This is because distinct regions are used for the fuel rods and the gas, and the rod thermal conductivity is significantly higher than that of the gas.

The CFD temperature contours do not exhibit the same symmetries as the other simulations. That is because the CFD simulations include buoyancy induced gas motion, which shifts the hottest locations upward compared to the Stagnant-CFD calculations. That motion also causes the upper surface of each basket opening to be hotter, and the lower surface to be colder, than it is if the gas is stagnant. This causes a net transfer of heat from the lower to the upper fuel region, and makes the average temperature of the upper fuel region hotter than that of the lower.

Buoyancy induced gas motion in each fuel region is characterized by its Rayleigh number, which is the ratio of buoyancy to viscous forces. The Rayleigh number is

$$ Ra_{\text{MAX}} = \frac{g \beta (T_{\text{MAX}} - T_{\text{MIN}}) H^3}{\nu \alpha} $$

(3)

The maximum gas temperature difference in the upper fuel region is $T_{\text{MAX}} - T_{\text{MIN}}$, where $T_{\text{CLAD,MAX}}$ is the maximum clad temperature and $T_{\text{BASKET,MIN}}$ is the minimum basket temperature. The CFD simulation in Fig. 4c predicts $Ra_{\text{MAX}} = 96^\circ C$. The acceleration of gravity is $g$, and $\beta$, $\alpha$ and $\nu$ are, respectively, the gas expansion coefficient, kinematic viscosity and thermal diffusivity of the gas. These gas properties are evaluated at the film temperature, $T_{\text{CLAD,MAX}} + T_{\text{BASKET,MIN}}/2$. For the results in Fig. 4c $Ra_{\text{MAX}} = 1.24 \times 10^7$.

Canaan and Klein [13 and 16] experimentally and computationally studied natural convection heat transfer for a horizontal 8x8 BWR spent fuel assembly within an isothermal enclosure. They considered nitrogen and helium gas. For Rayleigh numbers less than $10^6$ the Nusselt number was independent of
Rayleigh number. This is considered the conduction regime because buoyancy induced motion does not affect heat transfer. For Rayleigh numbers greater than $10^3$, the Nusselt number increased proportional to $Ra^{1/3}$. This is the convection regime. Rayleigh numbers between these values constitute the transition regime. The Rayleigh number for the simulation in Fig. 4c is therefore in the convection regime. The conditions of the current work are somewhat different from that of Canaan and Klein because the enclosure temperature is not isothermal and the rod temperatures are affected by radiation heat transfer.

Figure 5 shows temperature versus radial distance from the cask center $r$ from all three fuel models. Results are presented for the upper (U) and lower (L) diagonal lines shown in Fig. 2. Figure 5a and 5b show results for nitrogen and helium gas, respectively. We note that for the ETC and stagnant-CFD simulations these diagonals pass through the hottest fuel locations. For the CFD model the hottest fuel in each assembly is above the diagonals.

For both gases and all three models, the temperature profiles are nearly identical for $r > 0.225$ m, which is outside the fuel/backfill gas region. The ETC simulations give higher temperatures within the fuel than the other two models. The ETC temperatures from the upper and lower diagonals are identical because that simulation does not include buoyancy induced gas motion. Its temperature gradient exhibits a discontinuity at the interfaces between the interior and gap regions. This is because it employs a much lower conductivity in the gap than in the interior.

The Stagnant-CFD simulation results are also identical along the upper and lower diagonals. Its profile shape exhibits flat regions in the relatively high conductivity rods, and larger gradients in the lower conductivity gas. For nitrogen, the maximum cladding temperature from the Stagnant-CFD simulation is 51°C lower than that predicted by the ETC model. For both the nitrogen and helium simulations, the temperature gradient in the interior of the ETC model is roughly the same as that of the Stagnant-CFD simulation. However, the ETC gradients near the wall (in its gap region) are significantly higher. This appears to be the primary contributor to the different peak temperatures predicted by these two models.

For the nitrogen gas simulations, the CFD model does not give the same results for the upper and lower diagonals. For $r > 0.07$ m the upper diagonal temperatures are hotter than those of the lower. For the helium gas, the CFD temperatures along the upper and lower diagonal are nearly identical, and they are nearly the same as the Stagnant-CFD results. The conductivity of helium is sufficiently high that gas motion does not significantly affect heat transfer.

![Figure 5: Normal conditions of transport temperature profiles calculated along bold line U and L in Fig. 2 versus Radial Distance from the package center, for Q = 800 watts/assembly and with (a) Nitrogen Backfill gas (b) Helium Backfill gas.](image)

![Figure 6: (a) Exploded view of the velocity stream functions in the basket. (b) Velocity profiles in Y direction for Nitrogen backfill gas.](image)

**Gas Flow Field** Figure 6a shows streamlines in the two of fuel regions from the nitrogen gas CFD simulation with $Q = 800$ W. The streamlines in these two regions are similar but they are not identical, especially on their left sides. The heated rods and the cool walls cause the gas to move upward in the center of the regions and downward at their sides. This develops two counter rotating vortices. In both fuel regions the vortex closer to the cask periphery (the right) is significantly larger than the one near the cask center. This asymmetry is caused by the non-isothermal walls. In contrast, simulations for BWR and PWR assemblies within uniform temperature enclosures [17, 23] show symmetric vortices.

Figures 6b shows the vertical component of gas velocity versus distance from the left side of the fuel region, $X$. The speeds are for horizontal lines midway between the 7th and 8th row of rods.
(counted from the bottom) in both the top and bottom fuel regions of Fig. 6a. In both fuel regions the flow moves upward with fairly low speed in a large portion of the interior. It moves downward with significantly higher speed in thin regions near the side walls, especially on the right (near the cask periphery). The maximum gas speed in the upward and downward directions are $S_{\text{MAX,UP}} = 3.44$ and $S_{\text{MAX,DOWN}} = 10.92$ cm/s, respectively.

The multi-peaked shape of the profile is caused by the faster speeds above the regions between rods compared to that above the rods themselves. The profile shapes for the helium (not shown) is very similar to that of nitrogen but roughly ten times smaller. The speed profiles for helium in the upper and lower regions are even more nearly identical than they are for nitrogen. The Reynolds number for this simulation is $Re_H = S_{\text{MAX}}H/\nu = 691$. It does not appear that turbulence modeling is necessary for this simulation because $Re_H < 1000$.

### Dependence on Q

Figure 7a is a plot of the maximum temperature difference $\Delta T_{\text{MAX}}$ versus fuel heat generation rate $Q$. Nitrogen and helium gas results are presented from both CFD and stagnant-CFD simulations. $\Delta T_{\text{MAX}}$ is higher for nitrogen than it is for helium due to nitrogen’s lower thermal conductivity.

![Figure 7a](image)

**Figure 7:** (a) Maximum temperature difference between cladding and basket temperatures versus heat generation rate for CFD and stagnant CFD (b) Rayleigh number versus heat generation rate for helium and nitrogen backfill gases for CFD and Stagnant CFD simulations with heat transfer regimes predicted by Canaan and Klein, 1998.

For both gases, at low $Q$ heat is transferred from the rods to the enclosure primarily by conduction through the gas. This is because the temperature difference in the gas is too small to initiate buoyancy induced gas motion, and the surface temperatures are too low for radiation to be significant. As $Q$ increases the importance of gas motion and radiation also increase. This causes the slope of the $\Delta T_{\text{MAX}}$ versus $Q$ data to decrease.

For both gases the stagnant-CFD temperatures are hotter than the CFD results. This is because the stagnant simulations do not include the effect of gas motion. However, Fig. 7a shows that the effect of gas motion small. For nitrogen the difference reaches its highest level (2.3°C) at 800 watts/assembly. The differences are even smaller for helium.

Figure 7b shows the maximum Rayleigh number versus heat generation rate for He and $N_2$ from the CFD and Stagnant CFD simulations. Horizontal lines indicate the minimum Rayleigh number for the convection regime and the maximum value for the conduction regime [12].

The Rayleigh numbers for nitrogen are two orders of magnitude higher than for helium. This indicates that buoyancy forces are much more significant compared to viscosity for nitrogen than for helium. At low heat generation rates the Rayleigh numbers increase with the heating rate. This is because $\Delta T_{\text{MAX}}$ increases and this increases buoyancy effects. At higher $Q$, the gas temperature increases, and this changes its properties so that the ratio of buoyancy to viscous forces decreases. As a result, even though $\Delta T_{\text{MAX}}$ continues to increase (Fig. 7a), $Ra_{\text{MAX}}$ reaches a peak and then decreases (Fig. 7b).

For all heat generation rates considered in this work the helium simulations are deep inside the conduction regime [12]. This suggests that buoyancy induced gas motion has no effect on heat transfer. This explains why the Stagnant-CFD and CFD results for
helium are nearly identical in Fig. 7a. On the other hand, the nitrogen simulations are in the lower portions of the convection regime for heat generation rates between 250 to 1600 watts/assembly. This suggests that gas motion will enhance heat transfer compared to stagnant gas. However, the difference between the Stagnant-CFD and CFD results in Fig. 7a for nitrogen is small. This suggest that, even though gas motion does increase heat transfer compared to gas conduction and radiation, radiation heat transfer which is included in both simulations, dominates.

**Gas Speeds**

Figure 8a shows the magnitude of maximum upward and downward gas speed in the fuel regions versus assembly heat generation rate. Results for nitrogen and helium are presented. For all heat generation rates and both gases, the speed in the down flow is greater than that in the upward direction. Figure 6b shows that the downward flow is concentrated in thin regions near the enclosure walls and upward flow exists in a much larger portion of the domain interior. The gas speeds are larger for nitrogen than they are for helium, in agreement with the Rayleigh numbers presented in Fig. 7b.

The gas speeds increase with Q, reach a peak, and then decrease. The highest downward speed in nitrogen is observed at Q = 600 W. In Fig. 7b, the peak Rayleigh number for nitrogen is also observed at this heating rate.

Figure 8b shows the Reynolds number versus assembly heat generation rate. The Reynolds numbers are calculated using the downward velocities to obtain the maximum value. The maximum Reynolds numbers for helium and nitrogen are, 9.7 and 712 respectively.

**Maximum Allowable Heat Generation**

Figure 9a and 9b shows the peak (maximum) fuel temperature versus heat generation rate predicted by the ETC, stagnant-CFD and CFD models. Results for nitrogen and helium gas are presented in Figs. 9a and 9b, respectively. The peak temperature increases with the assembly heat generation rate for all three models and both gases. The ETC model predicts higher cladding temperatures than either CFD model. The stagnant-CFD and CFD results are so close together they are indistinguishable on this plot.

Figure 9c show the difference between the peak cladding temperatures predicted by different CFD models versus fuel heat generation rate. Two of the lines show the difference between the stagnant-CFD and CFD results for nitrogen and helium. The third shows the difference between the refined and nominal grids results for nitrogen between. The maximum difference between the refined and nominal grid is 2.3°C. This is 0.5% of the temperature difference between the hottest and coolest portions of the cask, and indicates that the current simulations are mesh-independent.

For nitrogen gas the peak cladding temperature predicted by the Stagnant-CFD calculation is 1.2°C hotter than the CFD value at Q = 600 watts. The difference is smaller at all other heating rates. For helium the difference is less than 0.5°C. The random nature and small size of this difference appears to be caused by numerical error of the calculation technique.

The horizontal lines in Figs. 9a and 9b show the maximum allowable clad temperature $T_{CL} = 400 \degree\text{C}$ [5]. The allowable heat generation rate $Q_A$ is the rate that causes the cladding to reach $T_{CL}$. The allowable heat generation rate for both gases and all three models are summarized in Table 1. The CFD and stagnant-CFD models predict nearly identical maximum cladding temperatures and they predict nearly the same value of $Q_A$ for both gases. This indicates that it may not be necessary to expend the computational resources required to calculate fluid motion when predicting the allowable heat generation rate. Both CFD models predict higher heat generation rates than those predicted by the ETC model.

![Figure 9](image-url)

**Table 1:** Average temperatures and Max. Temperature differences along the walls of a SNF assembly.

<table>
<thead>
<tr>
<th>Gas</th>
<th>$Q_A$ [watts/assembly]</th>
<th>Upper Basket</th>
<th>Lower Basket</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>CFD</td>
<td>Stagnant CFD</td>
<td>ETC</td>
</tr>
<tr>
<td>Nitrogen</td>
<td>2025</td>
<td>2025</td>
<td>1400</td>
</tr>
<tr>
<td>Helium</td>
<td>2500</td>
<td>2500</td>
<td>2000</td>
</tr>
</tbody>
</table>

*Copyright © 2007 by ASME*
by radiation for helium than for nitrogen and so its heat transfer rate is less affected by uncertainty in cladding emissivity.

Basket Wall Temperatures

Figure 11 is a plot of the basket wall temperatures at the allowable heat generation rate versus wall coordinate, s. Figs. 11a and 11b show results for nitrogen with \( Q = Q_a = 2025 \) W, and helium with \( Q = Q_a = 2500 \) W, respectively. The s-coordinate is the distance in Fig. 2 along the basket surface starting from the basket opening corner closest the package center and ending at the corner farthest from that location. Results for four different cords are presented, two from the upper basket opening and two from the lower. The results from different cords are not distinguished in the figure because they are close to each other.

The surfaces close to the cask center are hotter and exhibit a larger temperature gradient than the surfaces near the package periphery. The profiles exhibit small amplitude waves that are caused by local heat transfer variations due to the nearby fuel rods. For nitrogen (Fig. 11a) the wall temperature varies from 240 to 390°C while the variation for helium (Fig. 11b) is 250 to 396°C. For nitrogen the temperature profiles along the four cords are not identical due to the effects of natural convection. However, the temperatures at each s-location are within 15°C of each other. The variation is much smaller for helium because the helium motion has a much smaller effect on heat transfer.

The average and non-uniformity temperatures, \( T_{AVG} \) and \( \Delta T_{NC} \), for the upper and lower openings are presented in Table 1. The non-uniformity temperature is the difference between the maximum and minimum wall temperatures. Results are given for both nitrogen and helium at the allowable heat generation rate for that gas. The average surface temperature is higher for helium than nitrogen. This is because helium’s higher conductivity makes the difference between its surface temperature and the maximum cladding temperature smaller than it is for nitrogen. Helium’s higher thermal conductivity makes its temperature non-uniformity smaller than that of nitrogen. The differences between these results for the upper and lower cells are smaller for helium than it is for nitrogen. This is because the effects of natural convection are stronger for nitrogen than for helium.

**SUMMARY AND CONCLUSIONS**

Highly radioactive spent nuclear fuel is transported in thick walled casks. The fuel is placed in square cross-section openings of a basket structure inside the cask containment volume. That 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 conductive so 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 and 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 the fuel/backfill gas regions than ETC’s, but they require more computational resources. The current work demonstrates the use of a cask CFD model and attempts to determine which heat transfer effects are significant, and which may be reasonably neglected in order to reduce the required computational resources.

A two-dimensional finite volume mesh is constructed that accurately represents the geometry of a legal weight truck cask, including the fuel inside. CFD simulations are performed that calculate buoyancy driven gas motion, as well as the natural convection and radiation heat transfer in the gas filled fuel region. They also calculate conduction within the cask solid components. The cask is in a normal transportation environment. The fuel and cask temperatures are calculated for ranges of fuel heat generation rate and cladding emissivity, for both helium and nitrogen backfill gas. The maximum allowed fuel heat generation rate, which brings the peak fuel cladding to its temperature limit, is also determined. The results are compared to stagnant-CFD simulations, in which the gas speed is set to zero, to determine the effect of buoyancy induced motion. The allowable heat generation rate is 23% higher for helium than for nitrogen due to helium’s higher thermal conductivity. Increasing the cladding emissivity by 10% compared to the baseline value increased the allowed fuel heating rate by 4% for nitrogen, but only 2% for helium. It is higher for nitrogen because radiation transports a larger fraction of the heat when it fills the containment volume than when helium is present. The stagnant-CFD simulations give only slightly higher cladding temperatures than the CFD calculations. This is because buoyancy induced gas motion does not greatly enhance heat transfer compared to conduction and radiation for this configuration. The allowed heat generation rate from the stagnant-CFD calculation is therefore essentially the same as that from the CFD simulation. This suggests that future cask thermal calculations may not need to include gas motion. These results must be experimentally benchmarked before the CFD methods can be used with confidence for designing transport casks. Basket surface temperatures presented in this work can be used as the basis for boundary condition for those experiments.

REFERENCES


