A Comparative Analysis of Natural Convection Modeling Methods in a Horizontal Annulus and Its Application to Spent Nuclear Fuel Transfer Operations - 9206

J. Robles, C. Bajwa
United States Nuclear Regulatory Commission
Washington, DC 20555

E. Bagglieto
CD-adapco
Melville, NY 11747

ABSTRACT

Natural convection within an annulus bounded by two concentric horizontal cylinders, with a temperature difference across the annulus, has received considerable attention in past experimental and analytical investigations. This is due to the widespread applicability of this geometry to many disciplines including energy-conversion, energy-transmission, and radioactive materials storage and transportation. Experimental and analytical work in this area has developed different methods, approaches, and simplifications to model natural convection in these types of geometries. The purpose of the work described in this paper is to compare several different approaches to model this phenomenon (using computational fluid dynamics software) and to demonstrate how these approaches can be applied to an actual calculation of spent nuclear fuel (SNF) temperatures during dry cask storage operations.

Among the methods compared in this study are standard correlations, Large Eddy Simulation (LES), and Reynolds Averaged Navier-Stokes Simulation (RANS). The advantages and limitations of each of these methods will be discussed. In addition, the cases of isothermal and constant heat flux cylinders will be compared. Simplifications such as using two-dimensional rather than three-dimensional models will be compared, as well as the symmetry assumption and its applicability. The treatment of steady state and transient analysis will also be considered. Several previous experimental studies and results will be used to provide a benchmark for these methods and assumptions. Other studies will be discussed, and the validity of their approaches will be analyzed using this same benchmark.

Finally, a case study will also be presented that demonstrates the use of the developed approaches to model an actual situation. This case presents a generic spent fuel transfer cask that has an annular water-shielding cavity in the exterior and the fluid behavior within this region when subjected to constant heat flux from the spent fuel. The modeling approaches studied here will be compared with the standard annular cavity correlation approach for this case. There are significant modeling challenges, since the geometry of the cask must be balanced against computational limitations (such as mesh size and cpu time) and the need for accurate predictions of heat transfer through this fluid region. The diameter of the cask is much larger than the actual thickness of the annulus, meaning that the ratio of the outer to inner radius ($\kappa = r_o/r_i$) is close to one (1). This case

has not been studied as extensively as other cases (e.g. the case where $r_o/r_i=2.6$) and to the knowledge of the authors, no experimental data exists for this type of geometry. The main objective of this effort is to gain a broader understanding of the behavior of natural convection of a fluid in an annular region in order to select the suitable approaches for modeling this phenomenon in specific cases related to the analysis of SNF temperatures during dry cask storage transfer operations.

INTRODUCTION

Consider two infinitely long concentric hollow horizontal cylinders with inner radius r_i, outer radius r_o, and length L, as shown in **Figure 1**. The outer radius is κ times larger than the inner radius. The volume bounded by the cylinders is filled with a fluid substance. This volume is called an annulus. Numerous studies and experiments have been performed regarding natural convection in this type of geometry due to its relevance in many fields such as power transmission and nuclear power. This work focuses on spent nuclear fuel applications. Many spent nuclear fuel storage and transportation packages have a neutron shield that is in essence a fluid filled annular cavity.

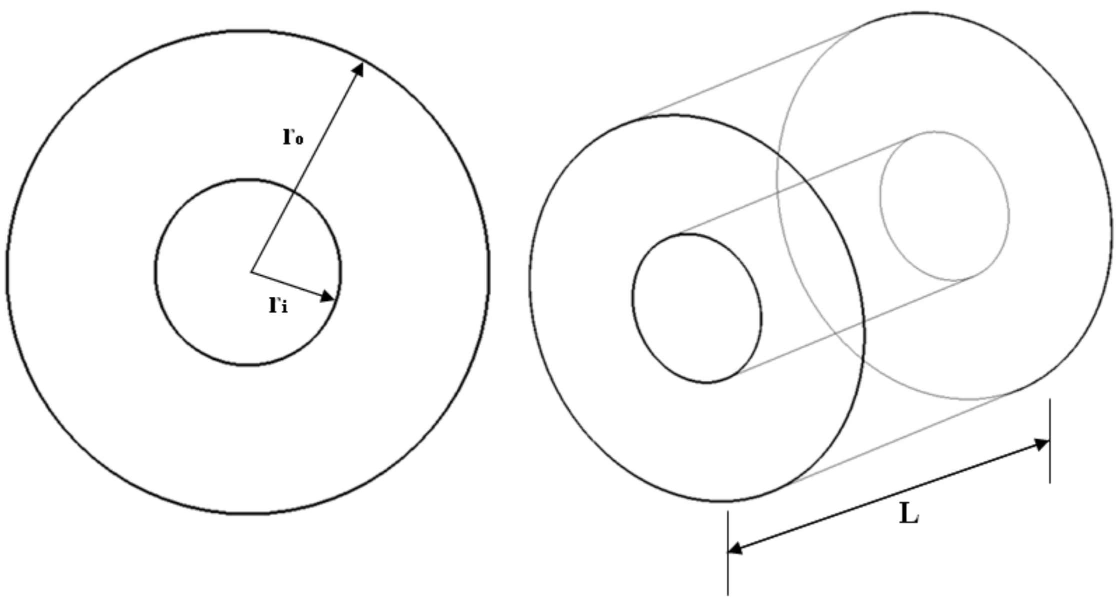

Figure 1: Annulus

Several numerical methods have been developed to describe the convective heat transfer in annuli. The Reynolds Averaged Navier-Stokes (RANS) method, Large Eddy Simulation (LES) method, and effective thermal conductivity methods are the main methods discussed here. All three methods are compared in order to determine which is most applicable to the thermal analysis of spent nuclear fuel storage and transportation applications. In addition, a simplified equivalent conductivity method is compared. Experimental data is used as a benchmark for these methods.

In addition to the different approaches discussed, several different assumptions and simplifications will also be compared. These include boundary conditions (constant heat flux versus constant temperature), symmetry, flow regime, transient or steady flow, and two or three-dimensional applicability.

Experimental Data

Kuehn and Goldstein [1] conducted an experimental study of natural convection within a concentric annulus with radius ratio (κ) equal to 2.6. In this experiment, the inner and outer wall temperatures of the annulus were kept at a constant temperature. The working fluid in this experiment was pressurized nitrogen. This data has been used as the benchmark for many numerical models including Desai & Vafai [2], Francis et. al [7] and others. The data ranges from $2.2\text{x}10^2$ to $7.7\text{x}10^7$. In this study, the plume rising from the top of the inner cylinder begins to oscillate with no discernable frequency at a Rayleigh value of $2\text{x}10^5$. The plume begins to become turbulent at a Rayleigh value of $1.6\text{x}10^7$ and the entire plume becomes turbulent at $2\text{x}10^7$. The rest of the cylinder remains virtually steady below these Rayleigh values. As the Rayleigh number approaches $8\text{x}10^7$, the region in the upper part of the inner cylinder also becomes turbulent. The bottom of the annulus remains virtually steady.

Castrejon and Spalding [3] conducted an experiment similar to that described in the work performed by Kuehn and Goldstein. In this experiment, they consider constant heat flux in the inner cylinder rather than constant temperature. The radius ratio used in their experiment is approximately 11.875. This study considered only transient convection, meaning that they observed the behavior of the fluid only until the onset of turbulence. Therefore, the results presented in their paper are for laminar convection in water.

In most spent fuel storage applications, radius ratios for concentric annuli approach one. To the knowledge of the authors, there is no comprehensive experimental data for cases similar to this geometry. The present work uses the aforementioned experimental results to validate and analyze approaches that can be applied to other geometries.

Approaches

Several different approaches are discussed in this work. The RANS method is discussed first. The next approach to be analyzed is LES. Finally, the applicability and limitations of using the equivalent thermal conductivity method (k_{eq}) are examined. Different assumptions are used for each method, and these are explained in the corresponding sections.

Reynolds Averaged Navier-Stokes Method

As the name implies, this method solves the RANS equations. It explicitly models the near wall flow in detail, as opposed to LES. There are several one- and two-equation models, although a form of the standard two-equation κ-ε (k-epsilon) model is used in the applicable models throughout this work.

Large Eddy Simulation Method

This method separates large- and small-scale motions, then uses a sub grid scale (SGS) model to approximate the small eddys near the walls and explicitly models the larger eddys.

Padilla and Silveira-Neto [8] performed an LES model of the same geometry featured in Kuehn and Goldstein [1]. This study focuses in the transition into turbulence (i.e., the Rayleigh number approaches, but is less than 2.51×10^6). This did not focus on the turbulent flow regime.

An LES model was also developed for the present study. The results agree well with both the experimental data from Kuehn and Goldstein and the results obtained by Padilla, et al. In contrast to the Padilla study, the present study focuses on turbulence in the plume area rather than transition. The results of this model are discussed

Equivalent Thermal Conductivity (K_{eq}) Method

The equivalent thermal conductivity method consists of determining a value of thermal conductivity that would allow the same amount of heat transfer as natural convection would. Mathematically, it can be expressed as

$$k_{eq} = \frac{Nu_{D_i}}{Nu_{D_{icond}}}$$

where Nu_{Di} is the mean Nusselt number for convection in an enclosure and Nu_{Dicond} is the mean Nusselt number for conduction in an enclosure. These values for equivalent thermal conductivity are found empirically. Using this equivalent conductivity, the convection problem is turned into a simple conduction problem, and the total heat transfer across the annulus can be calculated. This method is advantageous in that it provides a quick and simplified way of calculating the heat transferred across the annulus.

The drawback of using this method is that it does not locally model the behavior of the fluid in the annulus. There may be regions where the local heat transfer is much lower than in other regions. The models developed for this work show that the bottom region in the concentric annulus is one such region. Assuming that the heat transfer is uniform throughout the annulus may cause an incorrect calculation of temperature distributions, which in turn would cause an inaccurate and non-conservative calculation of fuel cladding temperature in spent fuel applications.

Assumptions and Approximations

Constant Heat Flux vs. Isothermal Cylinders

Castrejon and Spalding [3] treated constant heat flux in their work. In their experiment, they considered only the initial behavior of the convection. The flow is therefore laminar.

Kumar [6] studied the difference between these two cases. In his work the author found that for smaller radius ratios (i.e. $r_o/r_i = 1.2$), as Rayleigh values increase past the conduction flow regime the Nusselt number increases at a greater rate for constant heat flux than for isothermal cylinders. That is to say, the heat transfer is greater if the inner cylinder has constant heat flux applied rather than constant temperature at higher Rayleigh values ($Ra>10^4$). This effect is dampened as radius ratio increases. This suggests that under certain circumstances assuming isothermal cylinders may be more conservative than assuming constant inner heat flux. However, the most realistic case for spent fuel storage applications is that of axially and radially varying heat flux. This can have a significant effect on results, particularly when more heat flux is applied to a region of the annulus with less equivalent heat conductivity, such as the bottom of the annulus.

Symmetry Assumption

It is common practice to assume symmetry across a vertical axial plane when modeling because it reduces mesh size and cpu time. Although this assumption is correct for certain applications, it may not provide accurate results for more complicated cases such as when the flow is turbulent. As stated in the experimental data from Kuehn and Goldstein, as the plume becomes more turbulent, it oscillates with no discernable pattern and behaves unsteadily. Since this plume lies on the symmetry plane and it oscillates randomly, modeling half of the geometry will not provide accurate results.

Laminar vs. Turbulent flow

Determining whether the flow within an annulus is laminar or turbulent is of great importance in understanding the flow's behavior. It is evident from the experimental studies mentioned in this paper that the flow in an annulus is varied, depending on properties such as Rayleigh number, geometry (e.g. radius ratio, eccentricity), and even the region of the annulus. It is difficult to model turbulent and laminar flow simultaneously and obtain accurate results without running complex models.

The radius ratio affects the onset of turbulence, as demonstrated by Kuehn and Goldstein [4]. As the radius ratio decreases, the Rayleigh value for which turbulence begins decreases. More experimental data is needed to determine the onset of turbulence for geometries applicable to spent fuel applications, where radius ratios are closer to one.

The flow regime determines the appropriate assumptions needed to model the flow. The experimental data suggests that as the flow gets more turbulent, the symmetry and steady assumptions, as well as the RANS method, will not yield accurate results.

For the laminar flow model developed for this work, the symmetry assumption and two-dimensional approximation yields results that are in agreement with the experimental data obtained by Kuehn and Goldstein [1]. The isotherms obtained from this model are comparable to the interferograms from the experiment for this model. **Figure 2** shows the non-dimensional temperature distributions across the annulus at different angles. The results shown in this figure agree well with the experimental values as well.

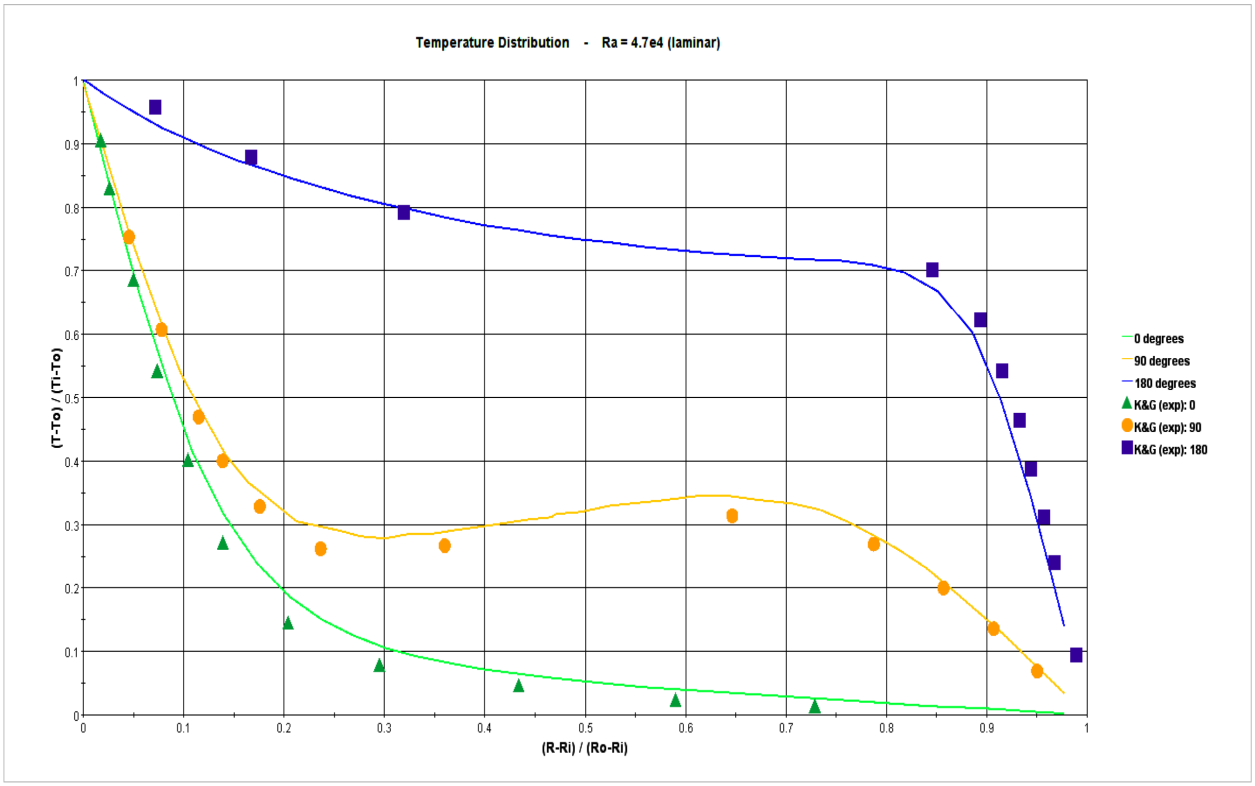

Figure 2: Laminar Temperatures Profiles (Ra = 4.7x10^4) for the two-dimensional, 1/2 symmetry model and Kuehn & Goldstein experimental data. Note: Angles are measured from the bottom (0°) to the top (180°) in all figures.

Transient vs. Steady

Desai et al. [2] created a model to approximate the Kuehn and Goldstein [1] experimental data assuming that the flow was steady. Francis, et al. [7] created a similar model using the same assumptions. Although certain areas of the annulus such as the bottom region behave as if steady, in the turbulent regime the hot plume rising from the top of the inner annulus is not steady. As noted by Kuehn and Goldstein, the plume oscillated with no apparent frequency or pattern. This disagrees with Desai's, et al. and Francis', et al. assumption that the behavior throughout the annulus is steady. Once the plume rising from the inner cylinder reaches turbulence, the behavior deviates from steady flow.

Two-Dimensional versus Three-Dimensional

As mentioned previously, a two-dimensional model of the annulus was developed for the laminar case (see **Figure 2**). This model provided results that are very close to the experimental data and saved computing time. Another model was created in two- and three-dimensions for the turbulent case. The RANS method was used for both models, as well as the half symmetry and steady assumptions. Although these assumptions were later demonstrated to be inaccurate in comparison with the Large Eddy Simulation method with transient and non-symmetry assumptions, this approach is useful to demonstrate that using a 2D approach will yield comparable results to the RANS method. The results of the model are shown in **Figure 3**.

It is evident that modeling this particular case in two dimensions yields comparable results to modeling in three dimensions. This is in spite of the length of the cylinder being short in comparison with the outer radius. It should be noted that this only demonstrates that the core region of the annulus (excluding regions near the end walls) can be modeled using a two-dimensional assumption only if the RANS method with steady assumption provides acceptable results, which is not the case when the plume is fully turbulent ($Ra = 2.51 \times 10^6$). **Figure 3** shows the temperature distributions for our RANS model with symmetry and steady flow assumptions. There is a clear over prediction of temperature, particularly in the upper portion of the annulus, near and at the plume location. As expected, the lower portion of the annulus remains steady as in the laminar case. This may lead to the incorrect calculation of heat transfer across the annulus.

For these turbulent flows, it is necessary to perform a LES in three-dimensions in order to fully understand the behavior of the fluid in the annulus. This type of case was run in order to model the annulus more accurately. The LES three-dimensional transient model results are shown in **Figure 4**, and they seem to agree better with the experimental data than the RANS approach used by Desai, et al. and Francis, et al.

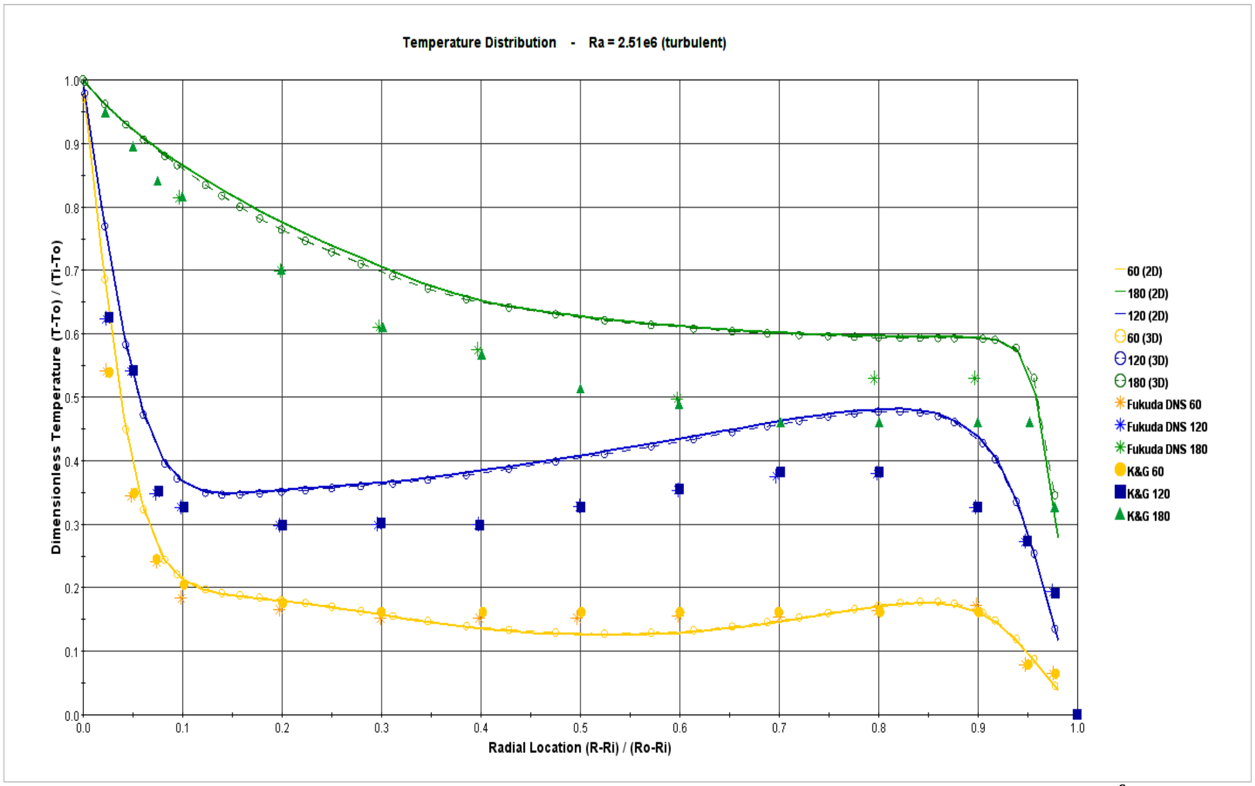

Figure 3: Temperature Profiles for RANS with steady and symmetry assumptions (Ra = 2.51×10^6) with experimental data. 2D and 3D data is also compared.

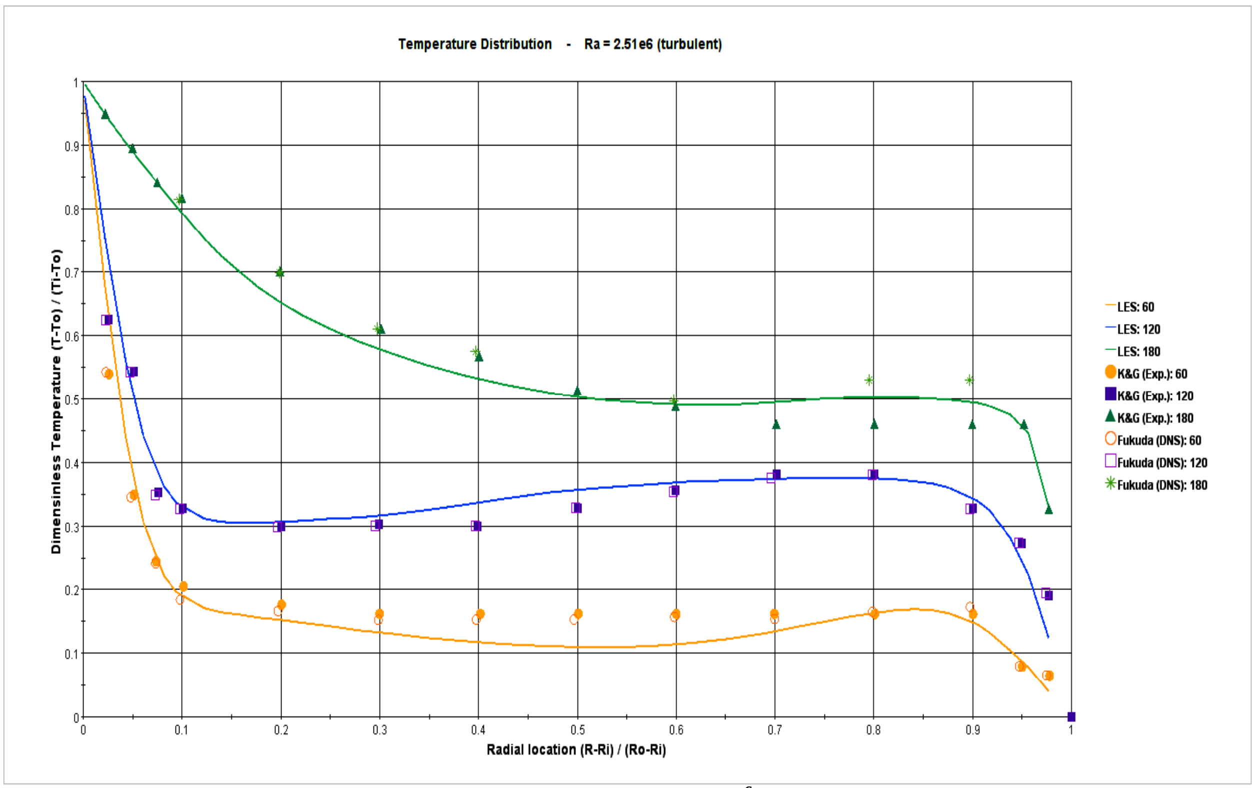

Figure 4: Temperature Profiles for LES (Ra = 2.51×10^6) with experimental data

An Applicable Annulus Model

A model of an annulus with a radius ratio of 1.12 was created to model a situation that is more applicable to spent nuclear fuel operations, although the results were not available at the time of this work's authoring. This model is limited because there is no experimental data for cases such as this, and there is uncertainty determining the transition from laminar to turbulent flow. However, it is useful in that it will demonstrate the behavior of the heat transfer across the annulus, which is essential in determining fuel cladding temperatures. The boundary conditions applied to this model are constant heat flux in the inner wall and convection on the outer wall. A convection coefficient based on heat transfer correlations for a horizontal cylinder is used on this outer surface.

The results of previous similar simulations agree with those of the 2.6 radius ratio models, in that there is a significant difference in heat transfer throughout the annulus. In the bottom region, there is a stagnant region of fluid. In the upper portion of the annulus, the hot plume becomes turbulent and thus allows for more heat transfer. This demonstrates that using the k_{eq} method may provide results that are not conservative.

Experimental data is needed in order to determine the validity of the results from this model. The transition from Rayleigh needs to be determined from this data, and the behavior of the fluid needs to be observed to determine whether it agrees with previous experimental data.

Conclusions and Recommendations

The behavior of a fluid in an annulus bounded by two concentric cylinders has been studied repeatedly. Several modeling methods were presented, including the RANS and LES methods. Assumptions such as symmetry, steady, and two-dimensional approximation were also presented. Several models were developed, and previous studies were discussed.

The results of the current and previous studies suggest that assumptions in thermal modeling have a great effect on the results obtained from simulations. Although these assumptions can simplify the model, they may lead to non conservative results. The two-dimensional, symmetry, and steady assumptions is valid for laminar flow in annuli. When the flow in the heated plume region becomes turbulent, a three-dimensional unsteady analysis is necessary. LES yields more accurate results in this region than the RANS model. The lower regions of the annulus behave much like the laminar steady models. This implies that there is less heat transfer in this region, and the temperature of the fuel cladding in the corresponding area may increase. Using the k_{eq} method does not consider this variation; consequently, the results may be inaccurate.

Lastly, experimental studies must be conducted in order to corroborate the results of the present work. Although data exists for larger annuli, further studies of different geometries that are applicable to spent nuclear fuel operations are needed.

References

1. T.H. Kuehn and R.J. Goldstein, An experimental study of natural convection heat transfer in concentric and eccentric horizontal cylindrical annuli, *ASME J. Heat Transfer* **100**, 635-640 (1978).
2. C.P. Desai and K. Vafai, An investigation and comparative analysis of two- and three-dimensional turbulent natural convection in a horizontal annulus, *Int. J. Heat Mass Transfer* **37**, 2475-2504 (1994).
3. A. Castrejon and D.B. Spalding, An experimental and theoretical study of transient free-convection flow between horizontal concentric cylinders, *Int. J. Heat Mass Transfer* **31**, 273-284 (1988).
4. T.H. Kuehn and R.J. Goldstein, Correlating equations for natural convection heat transfer between horizontal circular cylinders, *Int. J. Heat Mass Transfer* **19**, 1127-1134 (1976).
5. T.H. Kuehn and R.J. Goldstein, An experimental and theoretical study of natural convection in the annulus between horizontal concentric cylinders, *J. Fluid Mechanics* **74**, 695-719 (1976).
6. R. Kumar, Study of natural convection in horizontal annuli, *Int. J. Heat Mass Transfer* **31**, 1137-1148 (1988).
7. N.D. Francis, Jr., M.T. Itamura, S.W. Webb, D.L. James, CFD calculation of internal natural convection in the annulus between horizontal concentric cylinders, Sandia National Laboratories Report, 2002.
8. E.L.M. Padilla and A. Silveira-Neto, Large-eddy simulation of transition to turbulence in natural convection in a horizontal annular cavity, *Int. J. Heat Mass Transfer* **51**, 3656-3668 (2008).
9. K. Vafai and J. Ettefagh, An investigation of transient three-dimensional buoyancy-driven flow and heat transfer in a closed horizontal annulus, *Int. J. Heat Mass Transfer* **34**, 2555-2570 (1991).
10. J.K. Yoo, Dual free-convection flows in a horizontal annulus with a constant heat flux wall, *Int. J. Heat Mass Transfer* **46**, 2499-2503 (2003).
11. O.C. Zienkiewicz, R.L. Taylor, P. Nithiarasu, Finite Element Method for Fluid Dynamics, 6th ed., Elsevier, (2005).