Conference: 17th International Conference on Nuclear Engineering (ICONE 17)

Author1: Ghani Zigh
Author2: Jorge Solis
Company: US Nuclear Regulatory Commission

Tel 1: 301-415-5895
Tel 2: 301-492-3282
Fax: 301-415-5160

E-mail1: Ghani.Zigh@nrc.gov
E-mail2: Jorge.Solis@nrc.gov

Title: A Computational Study of Natural Convection Heat transfer in Concentric Horizontal Cylinder Annulus.

Natural convection in the annulus of two concentric horizontal cylinders is analyzed using Computational Fluid Dynamics (CFD). Numerous applications associated with this type of geometry exist in spent fuel storage transfer casks and energy conversion. In spent fuel transfer casks, water is used as a liquid neutron shield and decay heat is removed passively. Accurate prediction of heat transfer in the neutron shield cavity is important part to the prediction of the peak cladding temperature and the thermal performance of the transfer cask system.

Part 72 of Title 10 of the U.S. Code of Federal Regulations (10 CFR 72) provides the regulatory requirements for spent fuel storage and transfer cask applications. Specifically, 10 CFR 72.236 (f) requires that the spent fuel storage cask must be designed to provide adequate heat removal capacity without active cooling systems. In order to satisfy this requirement, the peak cladding temperature is limited to 400°C during storage and transfer, to avoid fuel cladding deformation caused by excessive creep and to limit the amount of radially oriented hydrides.

In the present study, a CFD method is validated using experimental data for Rayleigh numbers spanning both laminar and turbulent regimes. The Rayleigh number based on the gap width varied from $Ra_L=3.46x10^5$ to $Ra_L=1x10^9$. Additionally, this paper compares and discusses results of different turbulence models, including standard k-e model, low Reynolds k-epsilon model and transitional SST k-w model. These investigations were performed using two dimensional models. The local Nusselt number for the inner and outer cylinders as well as the fluid temperature in the gap obtained from the CFD prediction, experimental work and Direct Numerical Simulation (DNS) were compared. A good agreement was found between the results from the present investigation and previous experimental and computational works. The results from three dimensional analyses were also presented to investigate the effect of a realistic three dimensional flow characteristics. The obtained results demonstrated that if the annulus is sufficiently long, the core region over a substantial length of the cavity can be approximated by a two dimensional model.