3-D CFD SIMULATION OF A VENTILATED CONCRETE CASK USED FOR SPENT NUCLEAR FUEL STORAGE

A. Y. Walavalkar, D. G. Schowalter
Fluent Inc.
10 Cavendish Ct., Lebanon, NH 03766

ABSTRACT

Heat transfer performance of the VSC-17 system used for storage of spent nuclear fuel is modeled using the FLUENT commercial Computational Fluid Dynamics (CFD) tool. The VSC-17 system consists of a ventilated concrete cask (VCC) and a multi-assembly sealed basket (MSB). Heat generated by the spent nuclear fuel stored in canisters inside the MSB is transmitted through the MSB wall to the cooling airflow inside the VCC. Natural convection drives the cooling airflow through an annular gap between MSB and VCC. The geometrical description and experimental data were obtained for a cask performance test conducted at Idaho National Energy and Environmental Laboratory (INEEL), in conjunction with Pacific Northwest National Laboratory (PNNL), and reported by the Electric Power Research Institute (EPRI) [1].

A detailed model for a 90-degree section of the VSC system contains 878,000 cells, consisting of both hexahedral as well as tetrahedral cells. Simulation results for test conditions from the EPRI report are presented, including a comparison of computed temperatures to experimental data at various axial and radial locations throughout the system.

A validation study of this type establishes the importance of CFD simulations in the Nuclear Waste Management area. Such analyses can then be performed to understand safety issues, cooling patterns and overall heat transfer efficiency for cask designs at various heat loadings.

INTRODUCTION

For some time now, nuclear thermal hydraulics codes have been validated and used for waste transportation applications. In fact, part of EPRI's report on the VSC-17 system included validation of the COBRA code using the test measurements [1]. Other codes have been validated in a similar manner, including Westinghouse Electric's WECAN code [2], which took convection into account by using an effective thermal conductivity for the basket grid. Sophisticated couplings between codes have also been employed, notably between RELAP5-3D for convection, radiation and external heat transfer, and ANSYS for conduction [3].

These and similar methods have been employed for design purposes as well as for evaluating the impact of important scenarios such as fuel loading patterns [4] and severe fire accidents [5].

A major uncertainty that remains for system-level thermal hydraulic codes is in specifying convective heat transfer coefficients. Both experiment [6] and computational fluid dynamics [7] have been used to determine trends and correlations in this regard.

Only more recently have complete validation studies of Computational Fluid Dynamics tools been performed. An advantage of CFD codes is the direct simulation of flow and the resulting convective heat transfer to solid surfaces. Sakai *et al*. [8] showed that maximum concrete temperatures found in their CFD results agreed with test data. Heng *et al*. [9] made extensive comparisons of their CFD results to experiments for horizontal spent fuel storage casks and noted the importance of the choice of modeling laminar or turbulent flow, depending upon Rayleigh number.

The geometry of a cask storage system can be quite complex, from the spent fuel rods contained in numerous canisters, which are themselves generally stacked and enclosed in a larger basket with a gas such as helium or nitrogen for natural convection. This is usually surrounded by a larger concrete cask, with openings for natural ambient air circulation. While computational power is ever-increasing, the explicit resolution going from the level of the concrete cask all the way down to individual fuel rods probably requires more computational resources than most organizations are willing to employ. While the ultimate goal is generally to determine the highest fuel temperature so that cladding integrity is assured [10], it may be possible to use CFD in multiple steps, first by determining accurate heat transfer coefficients between the fuel rod containers and the surrounding gas, and then by modeling individual containers, explicitly resolving the fuel rods.

In the current study, we compare CFD results with the EPRI Report test data[1], focusing on the annular region between the outer concrete cask and the basket holding the fuel rod containers, as well as the volume inside the basket. We do not explicitly resolve the inside of the fuel canisters. The validation data include the effects of the MSB being pressurized with different gases at different pressures. The study shown here corresponds to the "vacuum" case, in which the MSB contained Nitrogen gas at –8.2 mbar gauge pressure.

METHODS

Geometry and Mesh Generation

The VSC-17 storage system consists of a VCC and a MSB. The VCC is a single piece concrete cylindrical vessel, which provides structural support, shielding and natural convection cooling for the MSB. The internal annular cavity of the VCC is formed by the gap between the inner steel cylindrical liner of the cask and the MSB outer shell. A flat plate at the bottom of the VCC provides support for the MSB placed inside the VCC. The air inlet and outlet vents of the VCC have non-planar paths to minimize the radiation escaping out of the VCC.

The MSB that is placed inside the VCC has an outer steel shell and an inner fuel guide sleeve assembly that holds the canisters containing spent fuel rods. The MSB is fitted with a shield lid that ensures MSB gas containment. Details of the VCC and MSB systems can be obtained from EPRI's report[1].

To create and discretize the geometry to be modeled, we used Gambit 2.1, a generalized unstructured geometry and mesh generation tool [11]. The geometry of the cask and MSB are described in the EPRI report, but detailed drawings which were necessary for a CFD model were supplied through personal communication with Pacific Northwest National Laboratory, with the permission of BNFL.

The VSC-17 geometry is symmetric about a 45-degree section. The canister heat loading, however, was not fully symmetric about this 45-degree section. With some approximation on the heat loading of the canisters, we could establish symmetry about the 90-degree section. Hence a 90-degree section of the VSC-17 geometry was discretized.

The canisters loaded with spent fuel rods were modeled as solids with volumetric heat generation sources. The approximate axial distribution of the volumetric heat generation profile was obtained from the EPRI report by curve fitting the axial decay heat profile.

The cooling airflow inside the VCC is driven by natural convection. Hence the inlet and outlet of the VCC were modeled with pressure specifications near both of these boundaries. The atmospheric pressure boundary condition was applied some distance away from the inlet and outlet to the cask, to allow for pressure variation across these regions.

Flow Solver

For computational modeling, the FLUENT 6.1 [12, 13] commercial CFD software was employed. FLUENT uses the SIMPLE algorithm [14] to solve the pressure, momentum, turbulence and convective heat transfer equations with a finite volume algorithm.

Based on the Rayleigh number calculation and previously published studies [9], it was determined that the flow inside the VCC was turbulent. A realizable k-ε model was used to model turbulence in the domain. The realizable k- ε model is a relatively recent development [15]. The term, "realizable" means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. Enhanced wall treatment was used for the near wall turbulence modeling, allowing a dynamic determination of whether to use a sub-layer or wall function approach. For more detailed information on the realizable k-ε turbulence model and the enhanced wall treatment as used in FLUENT, please refer to Fluent User Manual [12].

Radiation is solved using a discrete ordinates method [13, 16], in which the radiative transport equations are discretized into a finite number of solid angles and solved on the same finite volume mesh as the flow and convective heat transfer equations, coupled through appropriate source terms. For the present case, three divisions are used along the theta direction and three along phi.

On the outer wall of the VCC, two separate runs were made with different thermal boundary condition types. In most cases, the temperature on this outer wall will not be known and a heat transfer coefficient corresponding to natural convection will usually be imposed. For the first case, we have imposed a convection boundary condition with the convective heat transfer coefficient specified as 10 W/m^2K and the ambient temperature specified at 25 ˚C. The top and bottom covers of the VCC were specified as adiabatic and so was the bottom. In the latter case, a fixed wall temperature was imposed, using the data gathered in the field test.

The simulation was run in parallel on 2 processors on a 2 CPU Linux machine. The solution was progressed from an initial flow field by gradually increasing the value of the gravitational acceleration from a low initial value to final value of 9.81 m/s^2. This resulted in gradual development of the natural convection flow field. To aid convergence further, a first order upwind discretization scheme was used and a representative velocity field was applied at the cooling air inlet. Once the flow was developed, this was changed to a pressure inlet boundary condition. Also during initial iterations, the energy equation was under-relaxed significantly. This helped in stabilizing the solution process. The final converged steady state solution employed a second order spatial discretization scheme.

After the converged solution was obtained, the results of the simulation were post processed to compare the temperature values with those measured experimentally [1].

RESULTS

Comparisons with test data of temperature profiles from simulation at various axial as well as radial locations are presented here.

Figures 1-4 show the comparisons of the CFD simulation results with the experimental data. It can be seen that the CFD results match quite well with the experimental measurements. The deviation of the temperature rise above room temperature between the simulation and the experimental values is within about 10% at most of the locations except near the cooling air inlet and outlet. Where the temperature predictions are outside of this range, they are conservative (overpredicted), which is essential for safety analyses. Possible contributing factors to the discrepancy are the neglect of the steel plate liners in the inlet and outlet passages, limited discretization for the radiation model, the approximation of 90-degree symmetry, and the estimation of the decay heat profile. There was very little difference in the results between the case run with a fixed wall temperature and that run with a convective heat transfer coefficient for the outside of the concrete cask.

Figure 1 shows the comparison of the axial temperature profile predicted by the simulation with the experimental values at a radial location of 536.3 mm. This is inside the MSB shell. As can be seen from the comparison, the temperature predictions follow a similar trend as seen in experimental data.

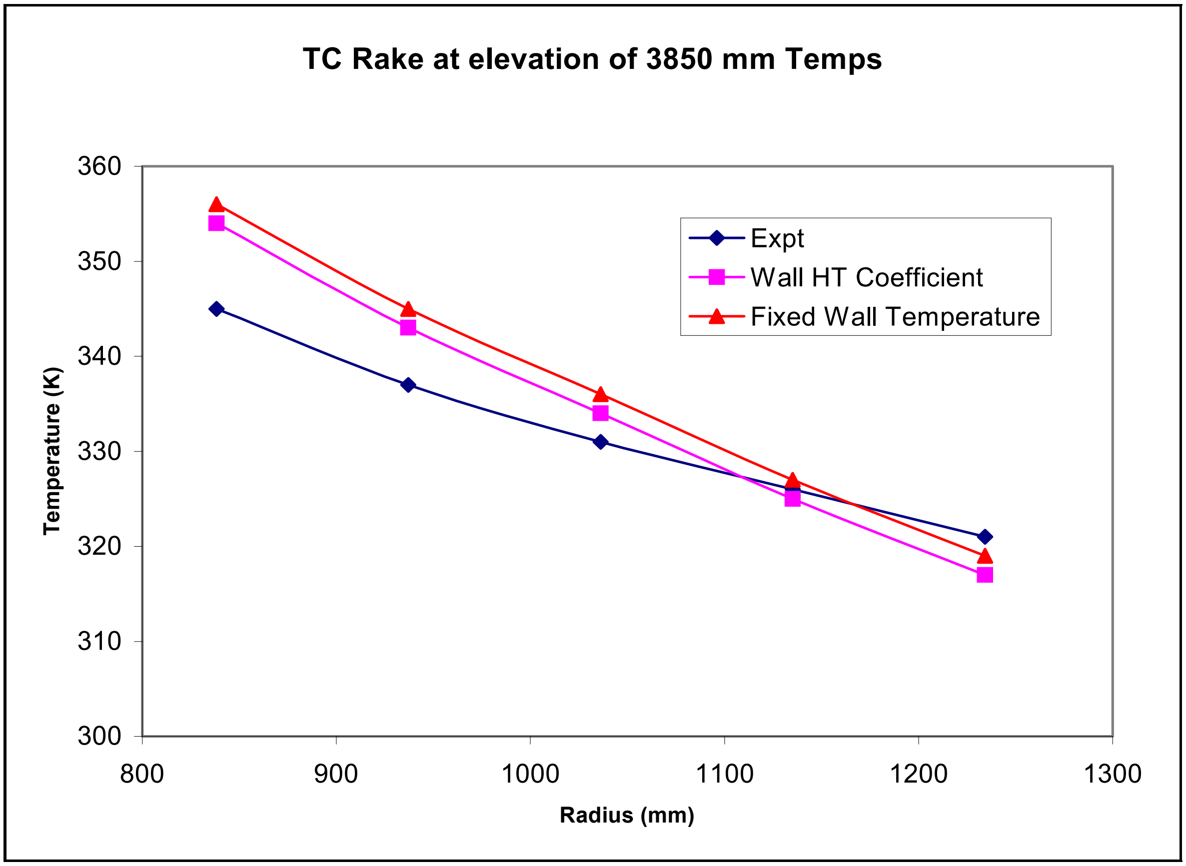

Fig. 1 Comparisons of axial temperature profile inside the MSB at a radius of 536.3 mm

Figures 2 and 3 show comparisons of wall temperature profiles on the MSB shell and VCC liner respectively. In these comparisons similar to the previous comparison, the trends and the temperature values represent the experimental data very well for most locations, except near the inlet and the outlet regions.

Figure 4 shows a comparison of the radial temperature profiles at an elevation of 3850 mm. All radial locations shown here are inside the thickness of the concrete outer wall of the VCC. Again, it can be seen that the trend as well as temperature values predicted by the simulation capture the experimental values with high accuracy.

Figure 5 shows contours of temperature on a plane through the center of the 90-degree section that was simulated. Overlaid lines show features of the geometry, including the outer concrete cask wall, along with inlet and outlet passages, and the MSB wall and MSB basket structure. Note that the outer rectangular (top) and semicircular (bottom) regions in the plot lie outside of the cask and allow for a constant pressure boundary condition far from the cask outlet and inlet, respectively. Sharp gradients at solid boundaries are also evident, along with the cooling effects of natural circulation, particularly in the gap between the inner cask wall and the MSB.

One concern upon close inspection of these results is that the vertical thermal gradients from the analysis seem to be smaller than those represented in the test data. In the simulations discussed above, the thermal conductivity of the canisters was assumed to be constant, and was set to a value of 220 W/m-K, a value typical of metals. In reality, of course, the canisters are filled with a combination of gas and fuel. One more simulation was run in which this conductivity was set to a value of 1 W/m-K, closer to typical gas values, but still larger to account for the considerable conductivity of the fuel. The results for this limited conductivity are shown in Fig. 6, along with the experimental values and those for the higher conductivity. At lower elevations, the agreement is superior for the lower conductivity, though it does not improve the results near the top of the canister.

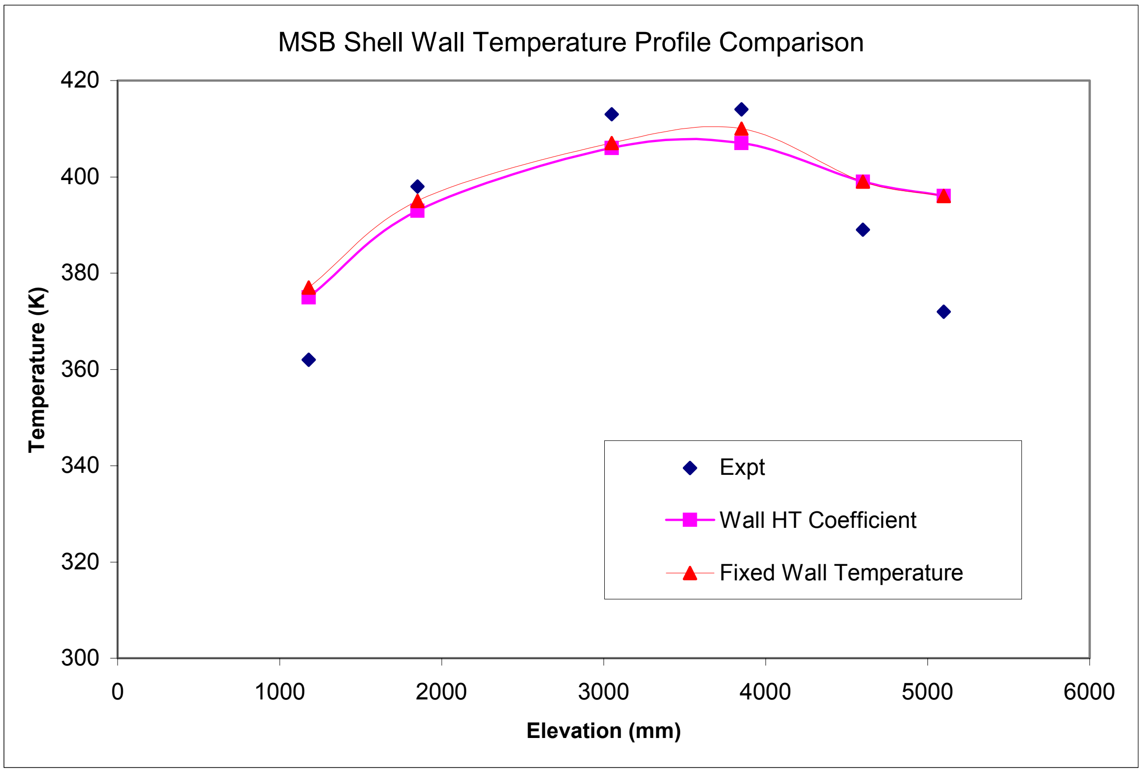

Fig. 2 Comparisons of wall temperature profile at MSB shell wall

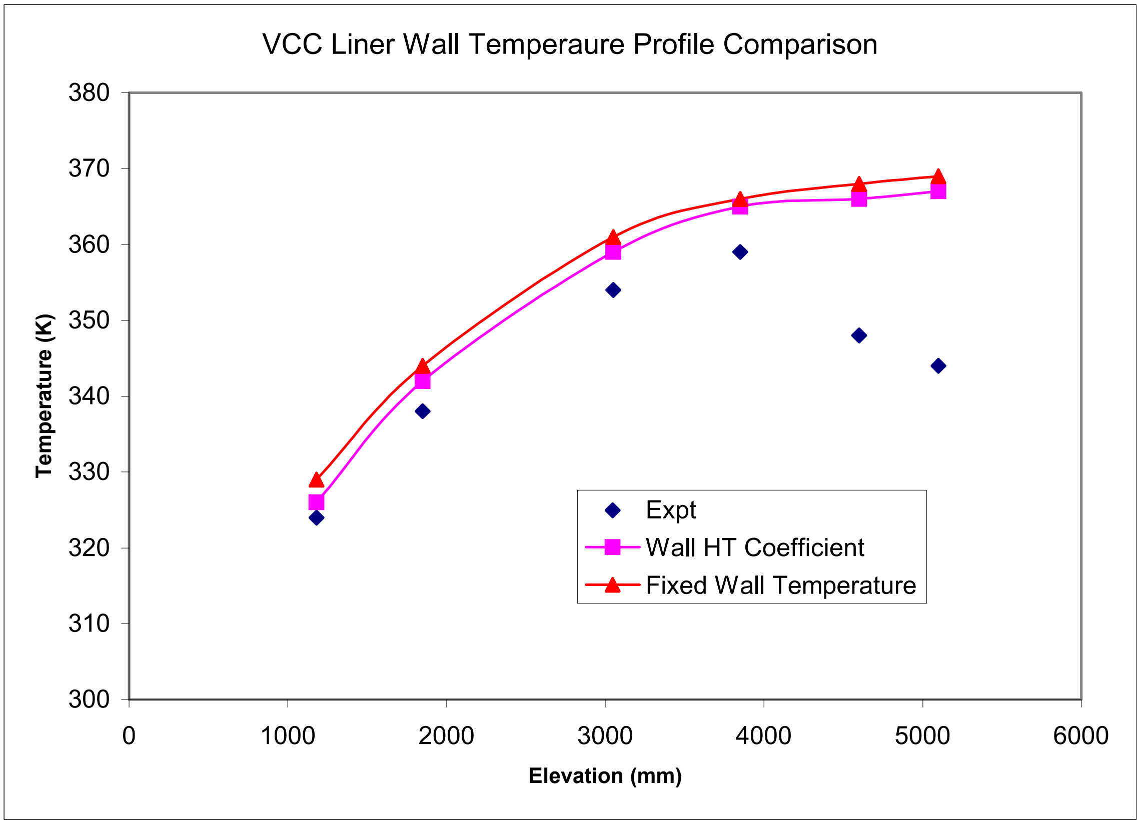

Fig. 3 Comparisons of wall temperature profile at VCC liner wall

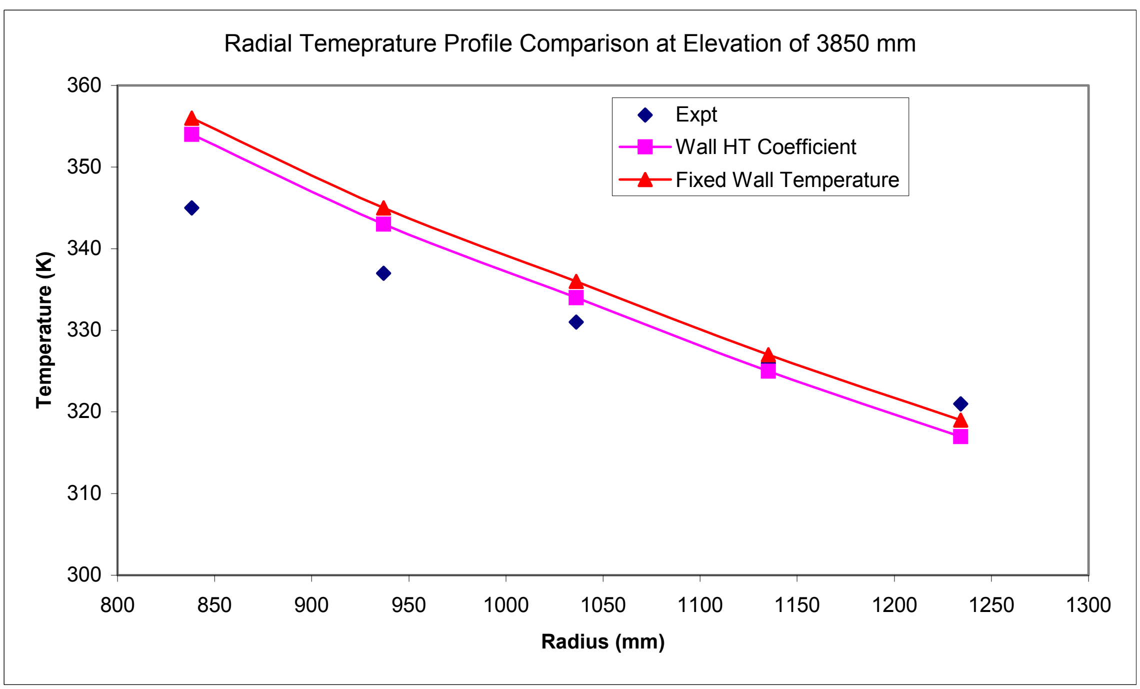

Fig. 4 Comparisons of radial temperature profile at an elevation of 3850 mm

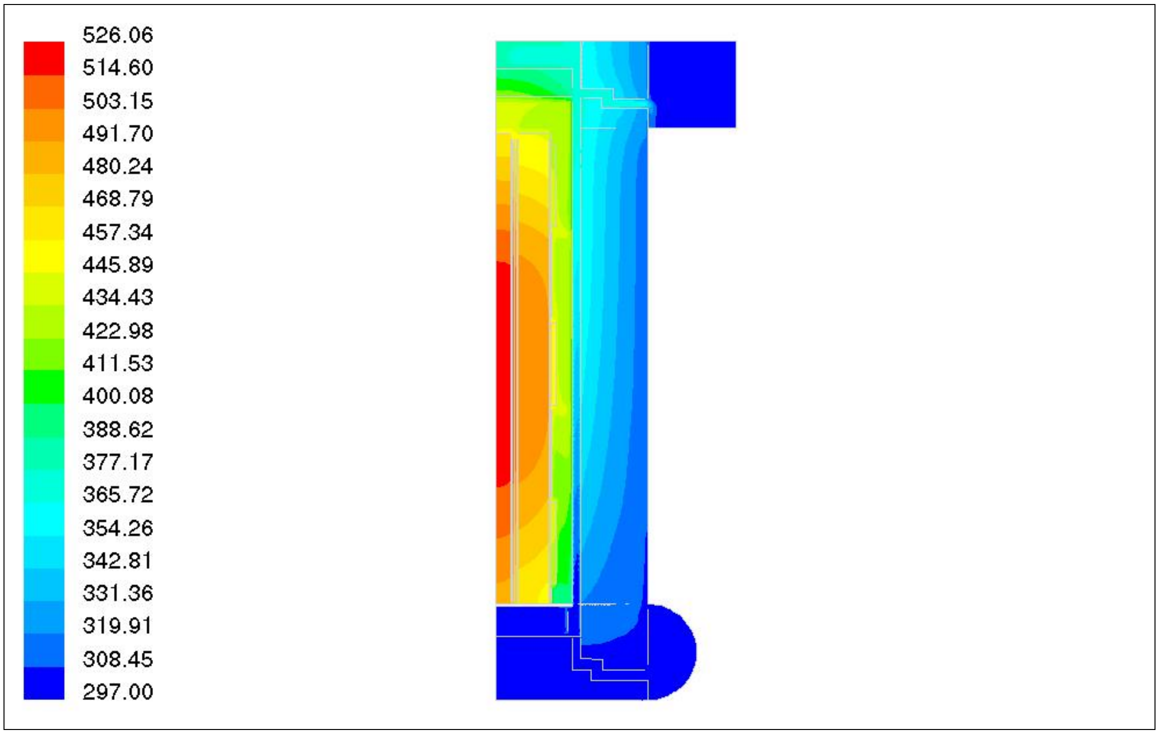

Fig. 5 Contours of temperature (degrees Kelvin) on a plane through the center of the 90-degree section.

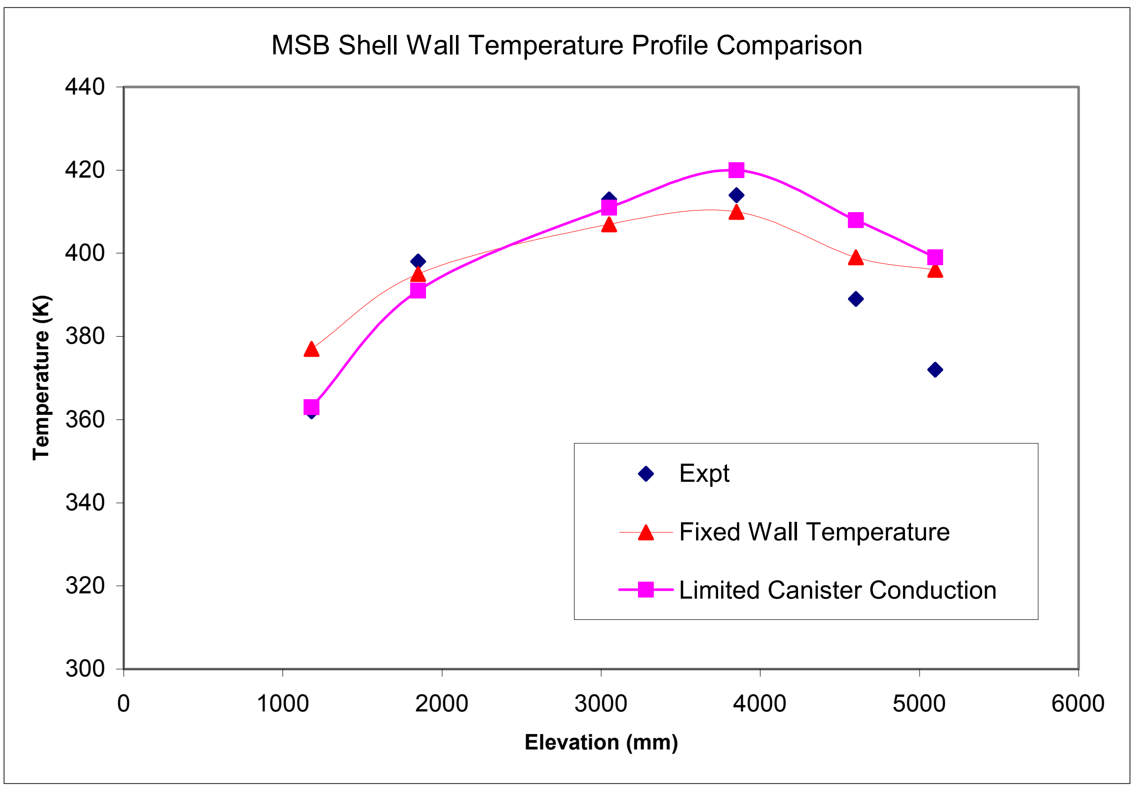

Fig. 6 Comparisons of wall temperature profile at MSB shell wall, including a case with a lower canister conductivity in order to account for low conductivity gas in the canister

CONCLUSION

Numerical simulation using the commercial CFD software FLUENT was performed for the VSC-17 spent fuel rod dry storage system. Flow equations with turbulence and energy equations with thermal radiation were solved for a 90-degree section of the VSC-17 system. Results of the simulation are compared with experimental data available. Comparisons of temperature profiles from simulation at various axial as well as radial locations with the experimental values are presented.

Simulation results are seen to predict the temperature values observed experimentally with high accuracy in most of the flow domain and conservative results elsewhere. This validation shows that CFD can be an effective tool in the Nuclear Waste Management area, providing much more detailed information about the flow field, leading to higher fidelity in heat transfer predictions and enabling further insight for design engineering. Such analyses can then be performed to understand safety issues, cooling patterns and overall heat transfer efficiency for cask designs at various heat loadings.

Planned future work includes looking at cases with different gases and pressures inside the MSB, as well as a careful investigation of the effects of including the steel liner at the inlet and outlet passages.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge the assistance of Ram Srinivasan, BNFL, Inc. and Dr. Mikal McKinnon, PNNL for their assistance with mechanical drawings of the MSB and storage cask.

REFERENCES

1 Pacific Northwest Laboratory and Idaho National Engineering Laboratory, 1992. Performance testing and analysis of the VSC-17 ventilated concrete cask. EPRI TR-100305.

2 J.Y. Hwang, R.P. Waszink, & L.E. Efferding 1991. Development of a thermal analysis model and design for a nuclear spent fuel storage cask. ASME 91-HT-4, presented at the National Heat Transfer Conference Minneapolis, MN, July 28-31.

3 F.-K. Ko, T.K.S. Liang & C.-Y. Yang 2002. Development of thermal analysis capability of dry storage cask for spent fuel interim storage. 10th International Conference on Nuclear Engineering (ICONE 10), Apr 14-18, v 4. pp. 463-471

4 M.A. McKinnon, J.M. Cuta, U.P. Jenquin 2002. Effect of loading pattern on thermal and shielding performance of a spent fuel cask. 10th International Conference on Nuclear Engineering (ICONE 10), Apr 14-18 2002, Arlington, VA. pp. 243-250

5 C.S. Bajwa, 2002. An analysis of a spent fuel transportation cask under severe fire accident conditions. Transportation, Storage and Diposal of Radioactive (ASME Pressure Vessels and Piping Conference), Aug 5-9, Vancouver, BC, Canada. v449, pp. 1-5

6 M. Satishchandra Arya, M. Keyhani 1990. Convective heat transfer in a sealed vertical storage cask containing spent-fuel canisters. Nuclear Science and Engineering, v 105, n 4, Aug. pp. 391-403

7 M. Keyhani & L. Luo 1994. Numerical study of convection heat transfer within enclosed horizontal rod-bundles. Proceedings of the 5th Annual International Conference on High Radioactive Waste Management. Part 2 (of 4), May 22-26 1994, Las Vegas, NV. v 2. pp. 780-787

8 M. Sakai, T. Sakaya, H. Fujiwara, A. Sakai 2002. Development of the concrete cask horizontal transfer system. 10th International Conference on Nuclear Engineering (ICONE 10), Apr 14-18 2002, Arlington, VA, v 4. pp. 377-382

9 X. Heng, G. Zuying, & Z. Zhiwei 2002. A numerical investigation of natural convection heat transfer in horizontal spent-fuel storage cask. Nuclear Engineering and Design, v 213, n 1, April, 2002. pp. 59-65

10 C. Bajwa, U.S. Nuclear Regulatory Commission, personal communication.

11 Gambit 2 User's Guide, Fluent Incorporated, N.H., USA.

12 Fluent 6 User's Guide, Fluent Incorporated, N.H., USA.

13 S.R. Mathur & J.Y. Murthy, 1997. A pressure based method for unstructured meshes. *Numer. Heat Transfer*, 31(2):195-216

14 S.V. Patankar, Numerical Heat Transfer and Fluid Flow. Hemisphere, Washington, D.C., 1980

15 T.-H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu, 1995. A New k-epsilon Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation. Computers Fluids, 24(3):227-238.

16 J.Y. Murthy & S.R. Mathur, 1998. A finite volume method for radiative heat transfer using unstructured meshes. *AIAA* 98-0860, 36th Aerospace Sciences Meeting & Exhibit, Reno, NV.