~~OFFICIAL USE ONLY~~

SANDIA REPORT

Revision 0
Draft

Analysis of Spent Fuel Pool Flow Patterns Using Computational Fluid Dynamics: Part 1 -Air Cases

Draft Completed: May 2003

Prepared by
R. Chiffelle*
KC Wagner*
R. O. Gauntt

Sandia National Laboratories
P.O. Box 5800
Albuquerque, NM 87185-0748

Information in this record was deleted in accordance with the Freedom of Information Act, exemptions 2 + 5
FOIA- 2004-0226

*Innovative Technology Solutions Corporation
6000 Uptown Boulevard, NE Suite 300
Albuquerque, NM 87110

Portions Ex 5 + Ex 2

~~OFFICIAL USE ONLY~~

G/5

5

Executive Summary

In 2001, United State Nuclear Regulatory Commission (NRC) staff performed an evaluation of the potential accident risk in a spent fuel pool (SFP) at decommissioning plants in the United States [NUREG-1738]. The study was prepared to provide a technical basis for decommissioning rulemaking for permanently shutdown nuclear power plants. The study described a modeling approach of a typical decommissioning plant with design assumptions and industry commitments; the thermal-hydraulic analyses performed to evaluate spent fuel stored in the spent fuel pool at decommissioning plants; the risk assessment of spent fuel pool accidents; the consequence calculations; and the implications for decommissioning regulatory requirements. It was known that some of the assumptions in the accident progression in NUREG-1738 were necessarily conservative, especially the estimation of the fuel damage. Furthermore, the NRC desired to expand the study to include accidents in the spent fuel pools of operating power plants. Consequently, the NRC has continued spent fuel pool accident research by applying best-estimate computer codes to predict the severe accident progression following various postulated accident initiators. The present report is Part 1 of a two part three-dimensional computational fluid dynamics (CFD) study to examine the flow patterns above, through, and around the spent fuel racks during accident conditions.

Ex 5

Ex. 5

Table of Contents

Table of Tables

Ex 5

Table of Figures

Ex 5

Ex 5

Ex. 5

Analysis of Spent Fuel Pool Flow Patterns Using Computational Fluid Dynamics: Part 1 -Air Cases

1 Background

In 2001, the NRC staff performed an evaluation of the potential accident risk in a SFP at decommissioning plants in the United States [NUREG-1738]. The study was prepared to provide a technical basis for decommissioning rulemaking for permanently shutdown nuclear power plants. The study described a modeling approach of a typical decommissioning plant with design assumptions and industry commitments; the thermal-hydraulic analyses performed to evaluate spent fuel stored in the spent fuel pool at decommissioning plants; the risk assessment of spent fuel pool accidents; the consequence calculations; and the implications for decommissioning regulatory requirements. It was known that some of the assumptions in the accident progression in NUREG-1738 were necessarily conservative, especially the estimation of the fuel damage. Furthermore, the NRC desired to expand the study to include accidents in the spent fuel pools of operating power plants. Consequently, the NRC has continued spent fuel pool accident research by applying best-estimate computer codes to predict the severe accident progression following various postulated accident initiators. The present report is Part 1 of a two part three-dimensional CFD study to examine the flow patterns above, through, and around the spent fuel racks during accident conditions.

Ex. 5

In Section 1.1, a description of the key phenomena expected in a SFP accident is presented. Two types of SFP accidents will be described, air cases and partial water cases. The present report examines the response of the SFP and surrounding room to a complete loss-of-coolant inventory accident (i.e., an air case). The partial loss-of-coolant accident is also described to illustrate the differences in the accident progression. Next, Section 1.2 discusses the approach and role of CFD codes to analyze SFP accidents. A description of the SFP model is given Section 2 as well as a single assembly benchmark calculation. Sections 3 and 0 have the results of the calculations and the conclusions, respectively.

Ex. 5

Ex. 5

1.2 CFD Approach

Parts 1 (i.e., the present report) and 2 [Ross, 2003] of the CFD study use FLOW-3D® to predict the flow patterns in the SFP. FLOW-3D® is a general purpose CFD code that has been used previously to predict flow patterns in a SFP [Wagner, 2000]). It is a relatively fast running finite difference code that is well suited to evaluating flow patterns in a SFP with porous media structures. There are more sophisticated finite element CFD codes, such as FLUENT (used in NUREG-1726). However, for the intended application of benchmarking flow patterns for the MELCOR control volume code, the level of sophistication in FLOW-3D® is adequate.

Ex. 5

2.2 Single Assembly Test Case

A single assembly was constructed in a two-dimensional rectangular grid to test the ability of the software to capture the relevant thermal and fluid physics of the problem. The results are compared to a similar analysis performed using MELCOR [Gauntt, 2000].

2.2.1 Test Case Dimensions and Parameters

The dimensions and parameters selected for the single assembly are given in Table 3.

Table 3. Single Assembly Test Case Fluid and Mesh Parameters.

Parameter	Spent Fuel Assembly	Fluid (air)
Viscosity (420 K) [Pa-s]	-	2.37×10^{-5}
Density (420 K) [kg/m^3]	7800	0.84
Thermal expansion [K^{-1}]	-	0.0015
Specific Heat (420 K) [J/kg-K]	580	1017
Thermal Conductivity (420 K) [W/m^2-K]	13	0.0343
Density*Specific Heat [J/m^3-K]	100	854
[redacted]		
Drag coefficient [-]	14	-
Temperature [K]	403	403
Pressure [Pa]	-	0.0
Power [kW]	1.0	-
[redacted]		
Constant lower boundary air velocity [m/s]	-	0.67

Ex. 2

Ex. 2

A steady-state energy balance determined from the specified mass flow rate and power gives the expected temperature change as,

$$Q = \dot{m} C_P \Delta T \qquad (1)$$

Where Q is the power, C_P is the specific heat, ΔT is the change in temperature, and $\dot{m}$ is the mass flow rate. The mass flow rate is also given by

$$\dot{m} = \rho A \mathrm{v} \qquad (2)$$

Where ρ is the density A the cross-sectional area, and v the velocity of the fluid. Substituting into the energy balance and re-arranging for the temperature change yields:

$$\Delta T = \frac{Q}{\rho C_P A \mathrm{v}}. \qquad (3)$$

Based on the specified parameters given in Table 3, the temperature change of the air is:

Ex. 2

To further verify the accuracy of the software, a similar test problem was prepared for MELCOR and the pressure, temperature, and velocity profiles were calculated. The results from the MELCOR calculation are summarized in Table 4.

Table 4. MELCOR Steady-State Solution for a Single Assembly.

Ex. 2

Ex. 5

5 References

Boyd, C., "Predictions of Spent Fuel Heatup after a Complete Loss of-Coolant Accident," NUREG-1726, July 2000.

Collins, T. E., and Hubbard, G., "Technical Study of Spent Fuel Pool Accident Risk at Decommissioning Nuclear Power Plants," NUREG-1738, February, 2001.

Flow Sciences, Incorporated, "FLOW-3D, Version 8.0 User's Manual", 2002.

Gauntt, R. O. et al, "MELCOR Computer Code Manuals, Reference Manual, Version 1.8.5, Vol. 2, Rev. 2", Sandia National Laboratories, NUREG/CR-6119, SAND2000-2417/1.

Ross, K. W., et al., "Analysis of Spent Fuel Pool Flow Patterns Using Computational Fluid Dynamics: Part 2 – Partial Water Cases," Sandia National Laboratories, Informal Report, April 2003.

Wagner, KC, Dallman, R. J., and Annon, M.C., "Best-Estimate Evaluation Of Loss-Of-Coolant Inventory In A Nuclear Power Plant Spent Fuel Pool," *International Meeting on Best-Estimate Methods in Nuclear Installation Safety Analysis (BE-2000)*, Washington, DC, November, 2000.

Release