

~~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

~~XXXXXXXXXX~~  
~~XXXXXXXXXX~~  
~~XXXXXXXXXX~~

## 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

<b>EXECUTIVE SUMMARY .....</b>	<b>II</b>
<b>1 BACKGROUND .....</b>	<b>1</b>
1.1 Accident Scenarios for CFD Analysis .....	1
1.2 CFD Approach .....	6
<b>2 CFD MODEL DESCRIPTION .....</b>	<b>9</b>
2.1 CFD Fluid and Material Parameters .....	9
2.2 Single Assembly Test Case .....	11
2.2.1 Test Case Dimensions and Parameters .....	11
2.2.2 Test Case Results .....	13
2.3 SFP Model and Mesh Geometry.....	17
2.4 Rack and Spent Fuel Geometry and Parameters.....	20
2.5 Boundary and Initial Conditions.....	25
<b>3 SIMULATION RESULTS .....</b>	<b>27</b>
3.1 Guide to the CFD Results.....	27
3.2 Results for Case 1.....	32
3.3 Results for Case 2.....	47
3.4 Results for Case 3.....	62
3.5 Comparison of Cases 1, 2, and 3.....	77
<b>4 CONCLUSIONS .....</b>	<b>78</b>
<b>5 REFERENCES.....</b>	<b>80</b>

Ex  
3

## Table of Tables

Table 1.	Summary of Modeling Assumptions in NUREG-1726 and the Present Study. .	8
Table 2.	FLOW-3D® Fluid and Material Parameters.....	10
Table 3.	Single Assembly Test Case Fluid and Mesh Parameters. ....	11
Table 4.	MELCOR Steady-State Solution for a Single Assembly.....	12
Table 5.	Single Assembly Test Case Variable Parametrics. ....	13
Table 6.	Comparison of Test 1 FLOW-3D® Solution to MELCOR and Energy Balance Calculation.....	14
Table 7.	SFP FLOW-3D® Rectangular Mesh Dimensions.....	18
Table 8.	SFP FLOW-3D® Mesh Size Parameters. ....	18

Table 10.	Summary of the Ventilation and Decay Heat Boundary Calculations. ....	26
Table 11.	Probe Location and Assembly Power Summary.....	31

Ex  
5

**Table of Figures**

Figure 1.	Reactor Building for a Boiling Water Reactor with the Spent Fuel Pool.....	4
Figure 2.	Spent Fuel Pool Rack with BWR Assemblies.....	5
Figure 3.	Prototypical Spent Fuel Pool Loading in an Operating BWR.....	7
Figure 4.	Single Assembly Test 1 Pressure, Temperature and Z-velocity Contours at Steady State Conditions.....	13
Figure 5.	Comparison of the Temperature Response for Test 1 and Test 2 at the Outlet of the Assembly. ....	15
Figure 6.	Single Assembly Test 3 Pressure, Temperature and Z-velocity Contours at Steady State Conditions.....	16
Figure 7.	Single Assembly Test 4 Pressure, Temperature and Z-velocity Contours at Steady State Conditions.....	17
Figure 8.	Mesh Domain of SFP Simulation (X-direction or N-S Plane) .....	19
Figure 9.	Detail of the Three CFD Mesh Blocks (Y-direction or E-W Plane).....	20
Figure 10.	Layout of the SFP Rack Geometry and Power Output.....	21
Figure 11.	Cross-Section of the SFP Rack and CRB. ....	24
Figure 12.	Side View Showing the Locations for the Z-axis Cross-sections of Temperature and Velocity in the SFP (Y-direction or E-W View).....	28
Figure 13.	Top Down View of the SFP Showing the Y-axis Locations for the Cross-Sections of the Temperature and Velocity Plots. ....	29
Figure 14.	Side-view Showing the Locations for the Four Different Z-axis Cross-sections of Velocity Vectors (Y-direction of E-W View).....	30
Figure 15.	Locations of the Three Probes in the SFP. ....	31
Figure 16.	Location of Equipment Pathway Duct as Portrayed on the CFD X-and Y-axis Boundaries for Case 1.....	32
Figure 17.	Temperature and Z-axis Velocity Profiles of Case 1 SFP. ....	33
Figure 18.	Temperature at the Top of the Racks in the SFP for Case 1.....	34
Figure 19.	Velocity Magnitude Under the Racks of the SFP for Case 1.....	35
Figure 20.	Temperature Under the Racks of the SFP for Case 1.....	36
Figure 21.	Temperature Profiles Along the Y-axis for Case 1.....	37
Figure 22.	Z-axis Velocity Profiles Along the Y-axis for Case 1.....	38
Figure 23.	Velocity Magnitude Vectors in the Refueling Building for Case 1.....	39
Figure 24.	Temperatures in the Refueling Building for Case 1. ....	40

Ex  
5

Figure 29. Temperature Response at the Inlet and Exit of an Empty Rack Cell Location for Case 1. .... 45

Figure 30. Z-axis Velocity at the Inlet and Exit of an Empty Rack Cell Location for Case 1. .... 46

Figure 31. Blowout Panel Location on the Maximum X-axis Boundary for Case 2. .... 47

Figure 32. Temperature and Z-axis Velocity Profiles of Case 2 SFP. .... 48

Figure 33. Temperature at the Top of the Racks in the SFP for Case 2. .... 49

Figure 34. Velocity Magnitude Under the Racks of the SFP for Case 2. .... 50

Figure 35. Temperature Under the Racks of the SFP for Case 2. .... 51

Figure 36. Temperature Profiles Along the Y-axis for Case 2. .... 52

Figure 37. Z-axis Velocity Profiles Along the Y-axis for Case 2. .... 53

Figure 38. Velocity Magnitude Vectors in the Refueling Building for Case 2. .... 54

Figure 39. Temperatures in the Refueling Building for Case 2. .... 55



Ex  
5

Figure 47. Temperature and Z-axis Velocity Profiles of Case 3 SFP. .... 63

Figure 48. Temperature at the Top of the Racks of the SFP for Case 3. .... 64

Figure 49. Velocity Magnitude Under the Racks of the SFP for Case 3. .... 65

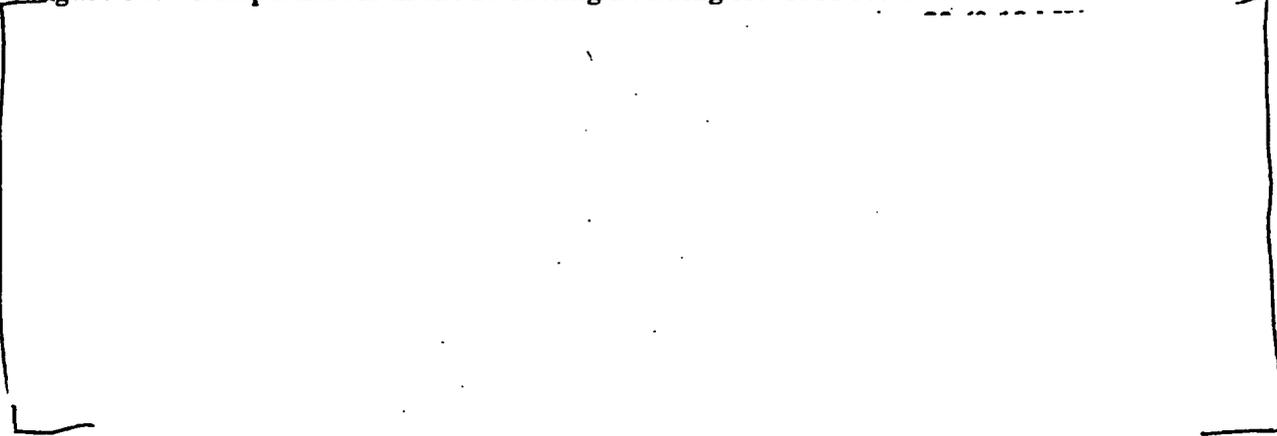
Figure 50. Temperature Under the Racks of the SFP for Case 3. .... 66

Figure 51. Temperature Profiles Along the Y-axis for Case 3. .... 67

Figure 52. Z-axis Velocity Profiles Along the Y-axis for Case 3. .... 68

Figure 53. Velocity Magnitude Vectors in the Refueling Building for Case 3. .... 69

Figure 54. Temperatures in the Refueling Building for Case 3. .... 70



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<sup>®</sup> to predict the flow patterns in the SFP. FLOW-3D<sup>®</sup> 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<sup>®</sup> 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]	-	$2.37 \times 10^{-5}$
Density (420 K) [kg/m <sup>3</sup> ]	7800	0.84
Thermal expansion [K <sup>-1</sup> ]	-	0.0015
Specific Heat (420 K) [J/kg-K]	580	1017
Thermal Conductivity (420 K) [W/m <sup>2</sup> -K]	13	0.0343
Density*Specific Heat [J/m <sup>3</sup> -K]	100	854
Drag coefficient [-]	14	-
Temperature [K]	403	403
Pressure [Pa]	-	0.0
Power [kW]	1.0	-
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,

$$Q = \dot{m}C_p\Delta T \tag{1}$$

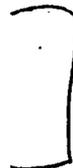
Where  $Q$  is the power,  $C_p$  is the specific heat,  $\Delta T$  is the change in temperature, and  $\dot{m}$  is the mass flow rate. The mass flow rate is also given by

$$\dot{m} = \rho Av \tag{2}$$

Where  $\rho$  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:

$$\Delta T = \frac{Q}{\rho C_p A v} \quad (3)$$

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