
AIR COOLED SPENT FUEL POOL HEAT UP PREDICTIONS USING COMPUTATIONAL FLUID DYNAMICS

Completed: January 2000

draft Dec 99

Prepared by
C. F. Boyd
Safety Margins and Systems Analysis Branch
Office of Nuclear Regulatory Research
Nuclear Regulatory Commission
Washington, DC 20555-0001

R/S

ABSTRACT

A generic study is completed to predict critical decay time for spent fuel in pool storage at nuclear power plants using computational fluid dynamics (CFD). Critical decay time is the minimum required age of the fuel in the pool such that a complete loss of coolant accident does not result in peak fuel clad temperatures rising above a critical temperature. The CFD model simulates the fuel heat up after a low probability complete loss of pool coolant accident. Air circulation is the primary means for fuel cooling. The study is completed using the FLUENT V5 computational fluid dynamics (CFD) code which is licensed for use by the Nuclear Regulatory Commission. A representative pool and containment building are used for the analysis. The grid used is constructed to provide the resolution needed to resolve the important phenomena while remaining small enough to provide efficient solutions. The overall size and computational complexity of the model preclude a rigid grid independence study. Quantitative results must be considered in light of this and other important modeling assumptions.

CONTENTS

<u>Section</u>	<u>Page</u>
ABSTRACT	iii
LIST OF FIGURES	vi
INTRODUCTION	1
COMPUTATIONAL FLUID DYNAMICS (CFD)	1
PHYSICAL MODEL	2
Fuel, Pool, and Racks	2
Containment Building and Ventilation	3
CFD MODEL	4
Finite Volume Mesh	5
Boundary Conditions	6
Material Properties	6
Fuel and Racks	7
Turbulence Modeling	7
Solution Convergence	7
RESULTS	4
CONCLUSION	7
REFERENCES	8

FIGURES

Description	<u>Page</u>
1. ½ of pool Region	2
2. SFSP and Containment Building	4
3. Finite Volume Mesh used for CFD Model	5
4.	10
5.	11
6.	11

TABLES

Description	<u>Page</u>
1. Fuel Loads	3
2. Material Properties (White, 1988)	6
3.	5
4.	10
5.	11
6.	11

INTRODUCTION

The Office of Research (RES) is evaluating post shutdown spent fuel pool heat up after a low probability loss of coolant accident using computational fluid dynamics (CFD). The CFD model is applied to predict the fuel heatup and natural circulation flow paths throughout the spent fuel pool and surrounding building. These predictions add insights into the air cooling phenomena which provides the bulk of the heat removal capacity during long term cooling scenarios under this sort of scenario.

Spent fuel pool heatup predictions are typically made using codes specifically tailored to the spent fuel rack geometry and/or physics. Codes of this type include SHARP¹, SFUEL, and COBRA-SFS. The sharp code manual (ref. 1) provides some general background information on some of the important phenomena involved in this type of model. A typical modeling approach involves the determination of one dimensional buoyancy-driven flows between upper and lower control volumes which tie the fuel bundles together. Codes such as COBRA-SFS also incorporate important physical models for conduction, radiation, flow losses, clad chemistry, and others. The assumption of a single well mixed volume which joins each of the bundles at the top of the rack structure, however, does not account for pressure or temperature variations across the racks. This upper component provides the ultimate heat sink for the model and is supposed to simulate the effect of the building and ventilation system. This simplifying assumption provides a needed boundary condition for the calculation and makes the problem tractable.

The calculations completed by RES use CFD to model an entire building and spent fuel pool. The assumption of a constant temperature, constant pressure component on top of the fuel racks is avoided. The CFD predictions do not include physical models for radiation, and clad chemistry. These models are considered important at elevated temperatures ($T > 600$ C, ref 2, Fig 12, 14, 26). The current predictions are geared towards low temperatures, where these effects are minimal. At elevated temperatures, the model is non-conservative. Simplifying assumptions are also made to model the complex geometry of the racks and fuel. A porous region represents the racks and fuel and provides an equivalent flow resistance for the bundles. A volumetric heat source adds the appropriate energy to the fluid in the active fuel region. The predictions are made for steady state conditions to predict the maximum fuel surface temperature for a given pool age and configuration. Previous studies indicate that during steady state conditions, the heat produced by the fuel is removed primarily through natural convection flows² (ref 2, pg 47). In addition, it is believed that the largest source of uncertainty in modeling these types of fuel pool accidents is the natural circulation flow rate³ (ref 3, pg 57). Considering the importance of the natural circulation flows, the fully 3 dimensional CFD predictions add valuable insights into this spent fuel cooling issue.

COMPUTATIONAL FLUID DYNAMICS (CFD)

The FLUENT⁴ CFD code is used to predict the temperature rise in the fuel and the natural circulation flow rates. Fluent is a commercially available general purpose CFD code for modeling fluid flow and heat transfer. The code solves the classic Reynolds Averaged Navier Stokes equation set on a finite volume mesh. Turbulence models are used to represent turbulent diffusion of momentum and energy. The meshing routines allow the model domain to

be adapted to almost any geometry. The CFD analysis involves the physical model description, the cfd model development, and analysis of the results.

PHYSICAL MODEL

A simplified model of the containment building, fuel pool, racks, and ventilation system are developed for this investigation. The dimensions for the buildings and other features were developed with input from the Office of Nuclear Reactor Regulation (NRR). Final dimensions are manipulated to simplify model development and improve model efficiency.

Fuel, Pool, and Racks

The pool is representative of a large BWR fuel pool. To take advantage of symmetry, it is centered on the long axis of the containment. Figure 1 shows the large features of the fuel pool region for $\frac{1}{2}$ of the pool cut along the line of symmetry. The finite volume cells used to represent the rack region in the CFD model are included in the figure to highlight the location of the racks and fuel. The overall pool dimensions are 9.54m x 11.08m and 12m deep. The fuel racks sit 0.15m (5.9 in.) from the pool floor and are 4.3 m (14.1 ft.) in height. The spacing between the fuel racks and the pool walls is also 0.15m. The high density racking occupies the entire lower pool (except the 0.15m gap around exterior). The square box racks share a wall between cells and there are no open cells or spacings between racks. The cell pitch is 0.154m (6 in). Looking down on the racks, this setup represents a single rack structure that is of 60 x 70 channels (9.24m x 10.78m).

Fuel. A fuel bundle is assumed to occupy each channel box of the high density racking (4200 fuel assemblies). The model assumes a uniform active (heat generating) length of 3.8 m (12.46 ft). The fuel is centered from top to bottom in the racks. Four regions are defined. A hot region with 800 bundles (f1 = 1 core), a second region with 267 bundles (f2 = 1/3 core), a third region with 267 bundles (f3), and a final region with 2866 bundles (f4). The hot fuel is assumed to remain in a contiguous

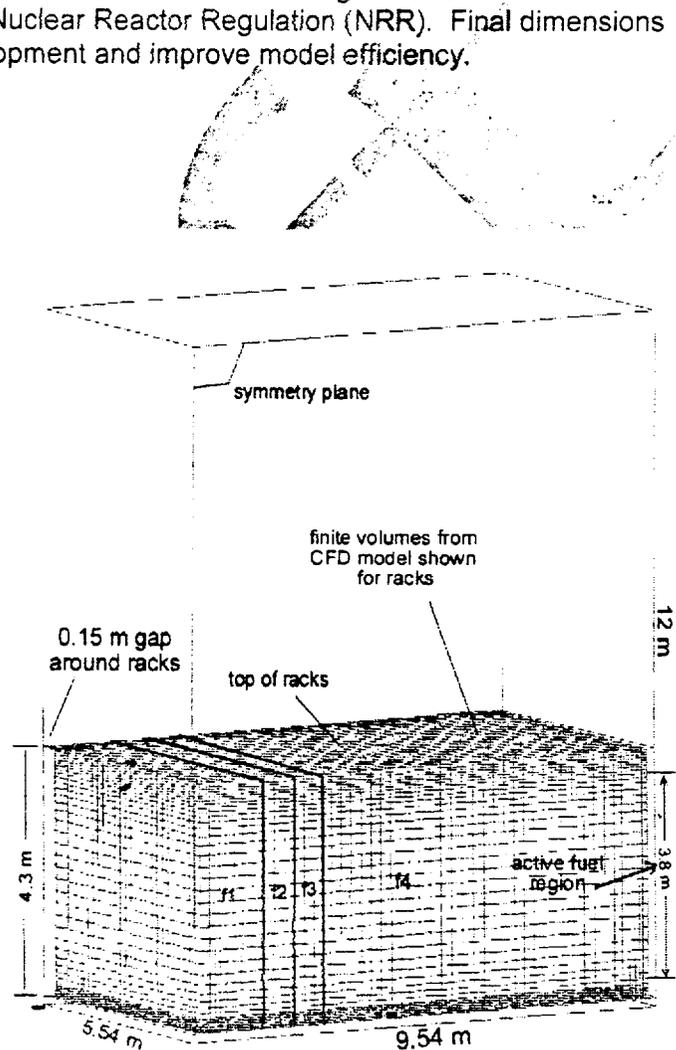


Figure 1. $\frac{1}{2}$ of Pool Region

region on the left hand side of figure 1. The next hottest 1/3 core is in another contiguous region right next to the hottest fuel. Next comes another 1/3 core. Finally, the remaining (oldest) 2866 bundles occupy the remaining region on the right hand side of the pool. More details on the geometry of the 9 x 9 bundles is given in Appendix A, Determination of Flow Resistance within Racks. Table 1 lists the fuel heat loads obtained from NRR⁵ for this study.

Table 1. Fuel Loads Used

Description	# cells	Watts/bundle after number of years		
		2	4	6
f1 Hot Fuel	800	866	418	307
f2 Med Hot Fuel	267	508	325	268
f3 Med Cold Fuel	267	367	277	246
f4 Cold Fuel	2866	310	253	231
Total heat load from pool		1.815 MW	1.220 MW	1.045 MW

The fuel loads are given in watts per bundle. These loads were obtained from NRR (ref 5) for this analysis and the details of this determination are not reported here.

Containment Building, and Ventilation

The containment building is sized to be representative of some BWR containment buildings. Since a specific building is not modeled, the final dimensions of the building are determined to make the CFD model setup straight forward and efficient. No internal structures within the building are included. One half of the building along with the fuel pool is shown in Figure 2. The plane of symmetry is facing out from the figure.

Ventilation. Two ventilation inputs (only one shown in figure 2 due to symmetry) are used for this building. Each ventilation opening is 4 square meters. The ventilation rate is set to approximate 2 building air exchanges per hour (Ref 2, pg 41). The ventilation is varied for sensitivity studies as noted in the results. The air is exhausted on the opposite end of the building through vents which mirror the input vents.

Wall Heat Transfer. The walls and ceiling of the building are assumed to transfer heat to the atmosphere. All other walls in the model are adiabatic. The overall heat transfer coefficient to the outside air is assumed to be in the range⁶ from 1 to 3 W/m²-K (ASHRAE, Handbook of Fundamentals, 1981). This range covers a variety of wall types with a 15 mile per hour outdoor air speed. For this study, overall heat transfer coefficients in the range of 0 to 4 are considered. The value used is considered uniform over the entire ceiling and building walls. Values used are noted in the results.

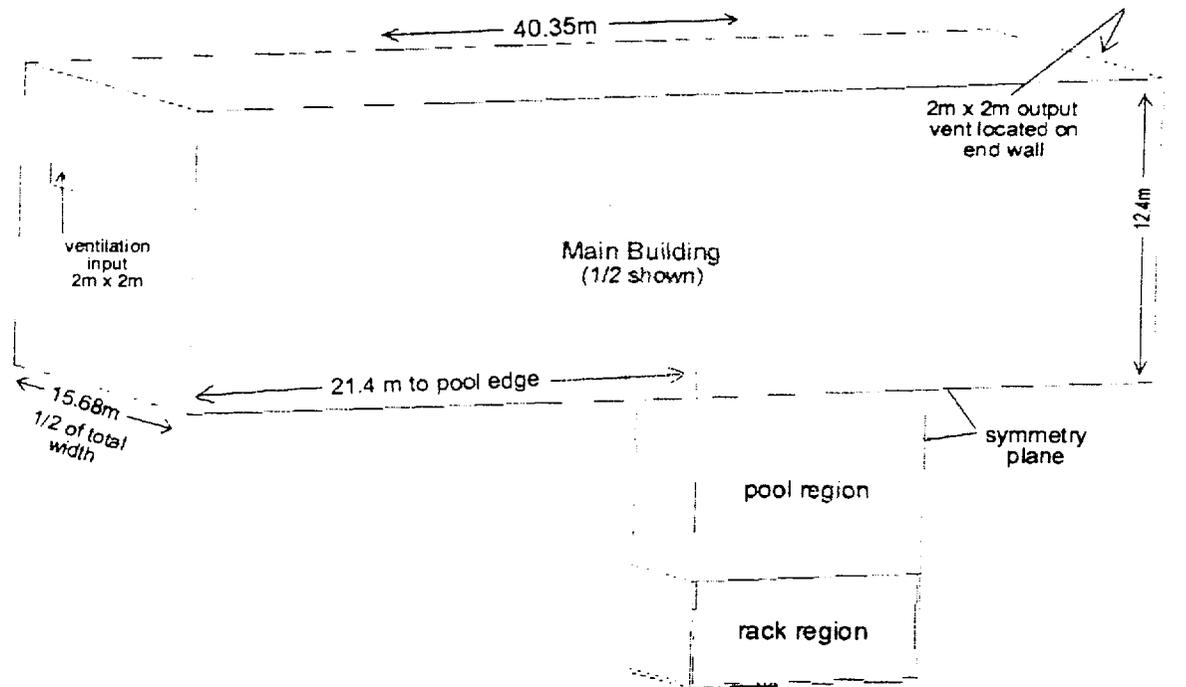


Figure 2. SFSP and Containment Building

CFD MODEL

The CFD model takes the physical model described above and represents it on a set of finite volumes over which the governing equations are solved. The Reynolds Averaged Navier Stokes Equations are discretized and approximated on the finite volume mesh. Many options in the CFD model can affect the final solution. A thorough investigation of these options is not feasible for the size of this problem and the amount of time available for resolution. The model is run in a steady state mode with sound CFD options. The solutions obtained highlight the maximum expected temperatures without an indication of the time it takes to reach this point. Completing a transient analysis would require a more detailed model and a significant increase in computer time. No models are used for the oxidation reaction or radiative heat transfer. The effect of these omissions is considered small at the lower temperatures. Higher temperature results should be viewed with this limitation in mind. A listing of the major options used for this analysis is given below and followed by more detailed descriptions of major features or assumptions.

- ▶ steady state Reynolds Averaged Navier Stokes solution
- ▶ k-epsilon turbulence model with standard wall functions
- ▶ variable material properties (air as working fluid)
- ▶ 675000+ finite volume cells (tetrahedron and hexagonal mesh)
- ▶ coupled solver with 2nd order differencing on momentum and energy
- ▶ porous media model of fuel and racks (vertical laminar flow through region)

Finite Volume Mesh

The mesh used for this problem consists of over 675,000 finite volumes and is shown in Figure 3. The mesh is designed to minimize the number of cells used for the model while providing an adequate number of cells to resolve important gradients in the flow. Cells are constructed to have minimal skew and equal length sides (aspect ratio = 1) wherever possible. Cell growth is limited to a range from 0 to 20 percent between neighboring cells.

Cells in the upper building are very large to save resources. Excess diffusion is expected in this area. This concession to save resources should not impact the final solution significantly. The flow in the upper building is believed to be more diffused in an actual facility due to structures within the building which are not present in the CFD model.

The majority of the cells are located in and around the fuel racks. The highest gradients and most important phenomena occur in these areas. Cell height at the top of the racks is close to 1 cm. This fine resolution is needed to resolve the large thermal and velocity gradients in this region. Transitions from the very fine mesh to regions with larger mesh are constructed using tetrahedral elements with a controlled growth rate. This technique reduces the skew in the hexagonal element regions and provides a smooth transition.

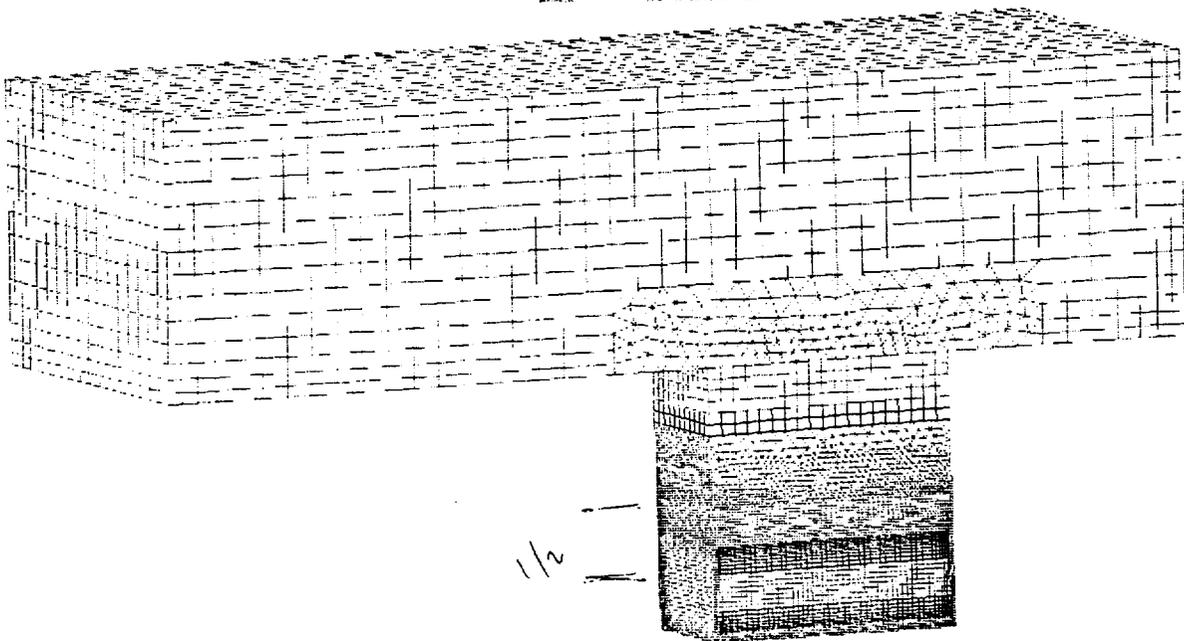


Figure 3. Finite Volume Mesh used for CFD Model

Boundary Conditions

The boundary conditions used include conditions at the walls (both internal and external) as well as conditions at the inlets and outlets. All external walls use the no slip condition for the momentum equations. No surface roughness is applied. The roof of the building and all sides are given a convective external boundary condition. The overall heat transfer coefficient is $2 \text{ W/m}^2\text{-K}$ (0 and 4 used for sensitivity study) with an external air temperature of 300K. The external walls of the pool and the floor of the building are considered adiabatic. The surface facing out from Figure 3 is modeled as a symmetry plane. Walls are used inside the fuel and rack region to separate zones of different fuel ages. These walls are given a no shear stress boundary condition since the flow resistance is accounted for by the porous model.

An inlet is created to carry fresh air into the building through a 4 m^2 opening (See Figure 2). At this location, the velocity is assumed normal to the building wall with a turbulence level of 10%. The volumetric flow rate at the inlet is equivalent to 2 building volumes per hour (1 and 3 building per hour considered in sensitivity analysis). The input air has a temperature of 300K. The resulting velocity at the inlet is 1.1 m/s. The air exits the building on the opposite end from the velocity inlet locations. The exit region is modeled as a pressure boundary which is set to atmospheric pressure (101325 Pa).

Material Properties

The working fluid is air. The air is an ideal gas with temperature dependent thermal properties which are given in Table 2.

Table 2. Material Properties (White, 1988)

Temperature K	Cp (J/kg-K)	k (W/m-K)	viscosity (Pa-s)
273	1003	.0242	1.72 e-05
373	1010	.031	2.17 e-05
473	1024	.0376	2.57 e-05
573	1045	.0439	2.93 e-05
673	1069	.0497	3.25 e-05
773	1093	.0553	3.55 e-05
873	1114	.0609	3.83 e-05
973	1135	.0659	4.09 e-05
1073	1153	.0703	4.34 e-05
1173	1170	.0747	4.57 e-05
1273	1184	.0786	4.79 e-05

Fuel and Racks

The fuel and rack structure are not be modeled in detail due to the complexity of this geometry. Modeling one bundle, in detail, using CFD is feasible with our current computer systems. Modeling every bundle in an entire pool is unrealistic. A porous media is used to represent the racks and fuel. The material does not affect the flow area in the fluent model but adds a user defined viscous and inertial flow loss. This loss is set to simulate the viscous and form drag losses through the racks and bundles. Details of this resistance determination are contained in Appendix A. The flow resistance in the lateral direction is set two orders of magnitude higher than the vertical direction to encourage vertical flow through the racks. Several vertical internal walls are included in the rack model to further ensure vertical flow. The fuel is centered vertically in the racks and has a uniform active heated length of 3.8 m. The heat is added to the flow as a volumetric heat generation term within the volume occupied by the fuel. Radiation is not accounted for.

Turbulence Modeling

The standard k-epsilon turbulence model is used for this analysis with standard wall functions. This model is selected because of its robustness and ease of application. It is acknowledged that the k-epsilon turbulence model is not specifically tailored to this type of problem. Experimental data is needed to refine the turbulence modeling beyond this standard approach. The flow up (or down) through the bundles of fuel is expected to be laminar due to the low Reynolds number in this region. To account for this, the porous medium region of the racks and fuel is modeled as a laminar region. In this region, the turbulence production and turbulent viscosity are disabled. All turbulent properties are transported through the region.

Solution Convergence

The solution is run using the steady state coupled fluent solver. Final solutions are not completely steady due to the nature of the problem. Solutions oscillated about a final solution. Convergence is monitored by plotting temperatures and velocities at various points in the domain. In addition, the residuals (a measure of error) are monitored and minimized for each solution. As a final check, an overall mass and energy balance is monitored.

RESULTS

A few sets of boundary conditions are applied to the given model to study the effect of various assumptions. Table 3 lists the completed analyses and the conditions associated with each. Parameters which are varied include the decay time, the ventilation rate, the overall outside heat transfer, and the flow loss coefficients in the racks.

Table 3. Analysis Descriptions

ID	decay time (yrs)	ventilation (bldgs/hr)	overall h (W/m ² -K)	sensitivity study notes
f4v2h0	4	2	0	4 yr case (base case)
f4v2h4	4	2	4	increased h
f4v3h0	4	3	0	increased ventilation
f4v1h0	4	1	0	decreased ventilation
f4v2h0v10	4	2	0	increased viscous loss +10%
f4v2h0i10	4	2	0	increased inertial loss +10%
f6v2h0	6	2	0	6 yr case

A primary goal of this analysis is the determination of the maximum fuel surface temperature as a function of the decay time. The 4 and 6 year results are completed to bracket the final critical temperature range. The sensitivity studies indicate the effect of various assumptions on the reported temperatures. Table 4 gives the maximum predicted fuel surface temperature for each of the completed cases.

ID	Tmax (°C)	delta Tmax (°C)	Notes
f4v2h0	900-1100°		
f4v2h4			
f4v3h0			
f4v1h0			
f4v2h0v10			
f4v2h0i10			
f6v2h0	660-910°		

REFERENCES

1. Nourbakhsh, H. P., et al, "Analysis of Spent Fuel Heatup Following Loss of Water in a Spent Fuel Pool, A Users' Manual for the Computer Code SHARP", NUREG/CR-6441, May 1998.
2. Benjamin, A.S., et al., "Spent Fuel Heatup Following Loss of Water During Storage", NUREG/CR-0649, March 1979.
3. Sailor, V.L., et al., "Severe Accidents in Spent Fuel Pools in Support of Generic Safety Issue 82", NUREG/CR-4982, July 1987.
4. "Fluent 5 Users Guide", Fluent Inc., Canterra Resource park, 10 Cavendish Ct, Lebanon, NH 03766, 1998.
5. Staudenmeier, Joe. , personal communications 1999.
6. ASHRAE, "Handbook of Fundamentals", 1981.

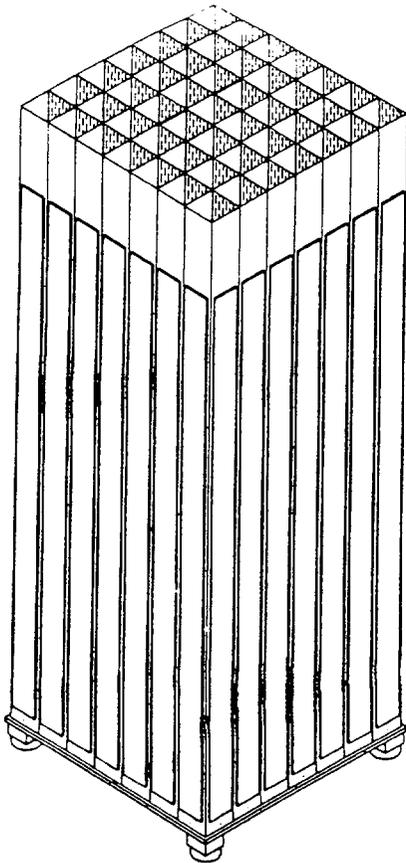
APPENDIX A FLOW LOSS COEFFICIENTS THROUGH BWR FUEL STORAGE RACKS

BACKGROUND

A computational fluid dynamics (CFD) analysis of spent fuel pool cooling is completed which models the spent fuel and racks as a porous media. This model requires an equivalent flow resistance which is input into the code to simulate the losses expected in the fuel and racks. The Fluent CFD code is used for this analysis. Fluent used a porous media model which acts as an added momentum sink in the momentum equations. The fluid does not accelerate through the medium since the volume blockage is not represented. The momentum source term is modeled with two parts. Equation A-1 gives the source term with the viscous (first term) and inertial loss terms for a one-dimensional case.

$$\frac{dp}{dx} = D \cdot \mu V + C \cdot \frac{1}{2} \rho V \cdot |V|$$

In 3 dimensions, the coefficients D and C represent matrices. These values are input into the Fluent code to represent the flow resistance. This appendix gives the background on the determination of D and C in the form required for the Fluent model.



GEOMETRY

Fuel bundles are stored in the spent fuel pool in racks which keep the bundles in an upright position and provide neutron absorption. A high density rack filled with 9 x 9 BWR fuel bundles is considered in this exercise. The bundle characteristics were obtained by RES from NRR. No reference for the information is given.

Figure A-1 shows a typical BWR high density rack. The square boxes are 0.154 m (6.06 inch) from center to center. Rack height is assumed to be 4.3m (14.1 ft) with an active fuel length of 3.8m (12.45 ft). The active fuel region is assumed to be centered in the rack from top to bottom. Rack distance from the floor is assumed to be 0.15m (5.9 inch). The walls separating the fuel cells are solid. For the purposes of the Fluent 3D CFD analysis, the racks are considered to be joined together to form a single rack of the type illustrated in Figure A-1 which spans the entire pool. The racks end a distance of 0.15m (5.9 inch) from each side wall of the pool. Each rack cell contains a complete 9x9 fuel bundle. With these assumptions, the racks form a homogeneous structure within the pool that can be described by considering a single rack cell.

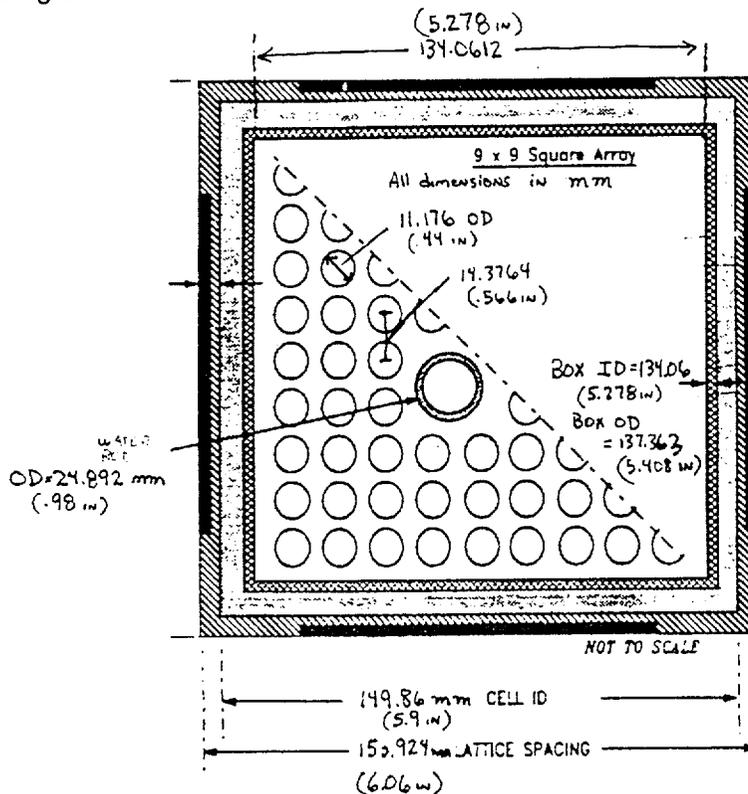


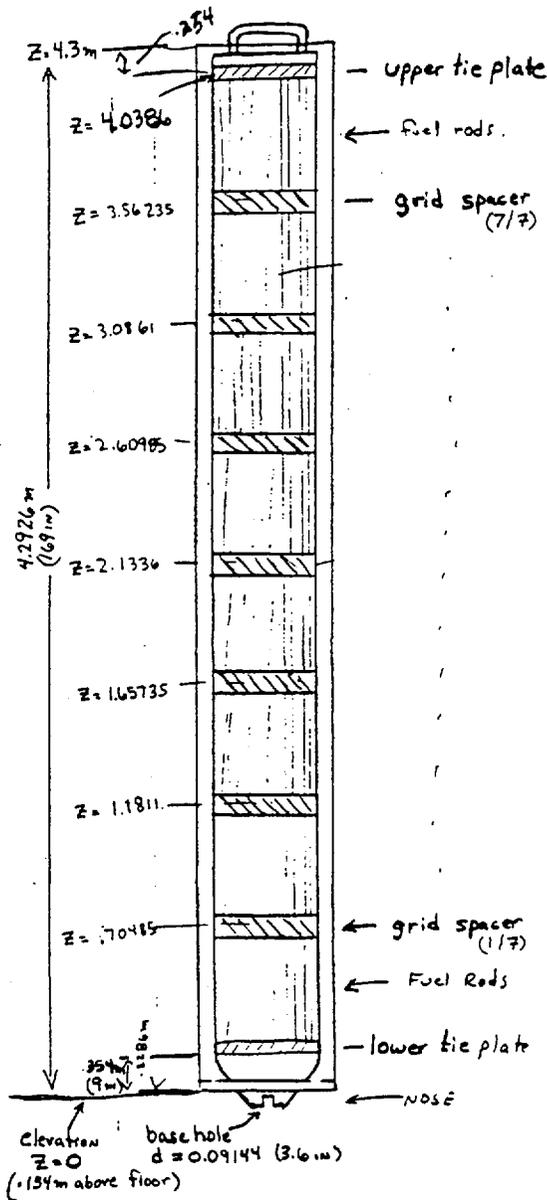
Figure A-2 shows a cross section of a 9x9 fuel assembly. The dark regions on each side represent a boron insert. The overall cell pitch is 0.154 m (6.06 in). Within the box, the fuel bundle is contained in a channel box with an inner dimension of 0.134m (5.3 in). A gap exists between the outer rack box structure and the inner channel box. For the purposes of this analysis, all fuel rods are considered to be full length rods. The rods have an OD of .1118m (.44 in) and a rod pitch of .1438m (.566 in). A total of 74 fuel rods and 2 water rods are included in this 9x9 cross section. Various values used to describe the geometry for the loss coefficient determination are given in Table A-1.

Table A-1. Values Used for Loss Coefficient Determination (1 rack cell)

D1	Flow area available to Fluent Calculation	$(0.153924)^2 = 0.023692597\text{m}^2$
D2	Channel Box Flow Area (no rods)	$(0.1340612)^2 = 0.017972405\text{m}^2$
D3	Channel Box Perimeter	$4 \times 0.1340612 = .5362448 \text{ m}$
D4	Area of Rods $(74 * \pi d^2/4)$	0.007259294 m^2
D5	Area of Water Rods	0.000973283 m^2
D6	Perimeter of all rods $(74 * \pi d) + (2 * \pi d_w)$	2.75457357 m
D7	Total Wetted Perimeter (D3 + D6)	3.2908 m
D8	Total Flow Area (D2 - D4 - D5)	0.0097398 m^2
D9	Hydraulic Diameter $(4 A / P) = 4 * D8 / D7$	0.0118388 m
D10	Flow area ratio at grid spacers	0.6 (area at spacers / area at tubes)
D11	Flow area ratio at tie plates	0.5 (area at tie plate / area at tubes)
D12	Lower orifice size (bottom of bundle)	0.09144m (3.6 inch)

LOSS COEFFICIENT DETERMINATION

The loss coefficients C and D must be determined for equation A-1. It is noted that the velocity in equation A-1 represents the Fluent velocity which is the flow through the unobstructed flow area (D1 from table A-1). In reality, the velocity will be higher due to the reduced flow area of the physical structure (D8 in Table A-1). Figure A-3 shows a single rack cell and fuel bundle arrangement. From this figure, it is noted that the flow will experience a viscous loss along the tubes and side wall as well as inertial losses at the lower orifice, the lower (and upper) tie plate, as well as the 7 grid spacers. Each of these losses is discussed below.



Viscous Loss

The viscous losses come primarily from the flow along the tube banks. It is assumed that the viscous losses are uniform for the entire length of the racks and the losses are based on correlations for tube flow. Equation A-2 defines the pressure loss over a distance L along the tube banks with hydraulic diameter d.

$$\Delta P_L = f \frac{L}{d} \left(\frac{1}{2} \rho V^2 \right)$$

The friction factor, f, is estimated¹ to be 96/Re. A value of 100/Re is used for this analysis. This is based on the actual flow along the tube bundles and the hydraulic diameter of the bundles. Using the Fluent velocity (V_F) which flows through the unobstructed region occupied by the rack cell and bundle results in Equation A-3.

$$\frac{dp}{dz} = 770231 \cdot \mu V_F$$

Comparing Equations A3 and A1 shows that the viscous resistance term for use in the Fluent

code is 770231.

Inertial Loss

Inertial losses come from the lower orifice (nose), the entrance into the tube region (at lower tie plate), the 7 grid spacers, and the upper tie plate. Values for these coefficients are determined from Reference 1 and from tables in Idelchik's book. At the low Reynolds numbers expected in the simulation, the loss coefficients vary and the uncertainty in these numbers is large. The effect is not large, however, because the inertial losses are small at low Re.

Table A-2 lists the parameters obtained to represent the inertial losses within the bundles. The k is the k factor obtained from the reference material. In some cases, interpolations or even extrapolations had to be made. V_r represents the reference velocity description associated with the k factor. k_F and V_F represent the friction factor and velocity based upon the Fluent model flow area (100% open area). The value C which is needed by the code is also given. Note that the effect of C in the code is not an abrupt pressure drop but an equivalent continuous pressure drop which occurs over the entire rack length L .

Table A-2. Inertial Loss Parameters

location	k	V_r based on	k_F	$C = k_F / L$
lower orifice	11	flow in open channel box	19.11	4.45
entrance to tubes	1.2	flow along tubes	7.1	1.65
grid spacers (1 of 7)	3.2	flow along tubes	18.94	4.4
upper tie plate (w expansion)	4	flow along tubes	23.67	5.5
effect of lower orifice, entrance to tubes, 7 grid spacers, and upper tie plate				42.5

Final Code Input Summary

The input parameters to the Fluent code to simulate the pressure drop through the rack and bundles are listed below.

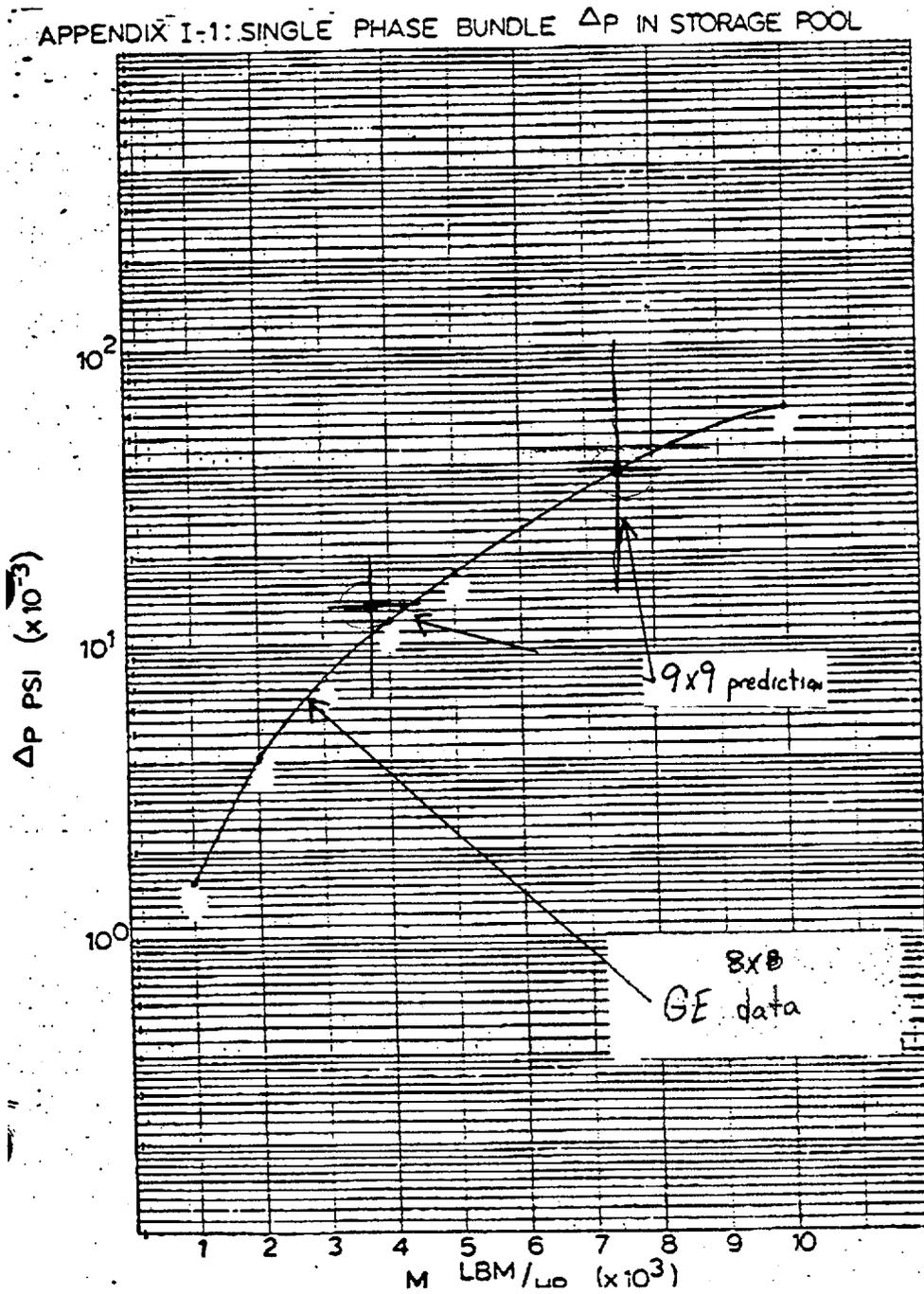
- ▶ viscous loss 770000
- ▶ inertial loss 42.5

These values are put into the code for the vertical direction. For the lateral direction, these values are multiplied by 100. This increase in resistance and the placement of solid wall in the domain limit the lateral flows.

COMPARISON TO GE DATA

A plot of pressure loss through and 8x8 GE bundle was obtained from a HOLTEC report and used to compare the results obtained above. The GE data represent the single phase pressure drop through a BWR bundle as a function of the flow rate. The results are mainly at low, natural circulation, flow rates similar to those assumed for this analysis. Although the geometry of the 8x8 bundle is not specified and is different from the 9x9 used above, these

results are considered a good indication of the quality of the above results. The pressure drop for 2 flow rates is computed from the coefficients given above and compared to the GE data on Figure A-4. The agreement is very good.



REFERENCES

-
1. Todreas, N. E., Kazimi, M. S., Nuclear Systems I, 1990.
 2. Idelchik