This document was prepared in conjunction with work accomplished under Contract No. DE-AC09-96SR18500 with the U.S. Department of Energy.

Thermal Evaluation of Contaminated Liquid onto Cell Floors

John E. McAllister, Jr., Ph.D., URS-Washington Division

Abstract

For the Salt Disposition Integration Project (SDIP), postulated events in the new Salt Waste Processing Facility (SWPF) can result in spilling liquids that contain Cs-137 and organics onto cell floors. The parameters of concern are the maximum temperature of the fluid following a spill and the time required for the maximum fluid temperature to be reached. Control volume models of the various process cells have been developed using standard conduction and natural convection relationships. The calculations are performed using the Mathcad modeling software. The results are being used in Consolidated Hazards Analysis Planning (CHAP) to determine the controls that may be needed to mitigate the potential impact of liquids containing Cs-137 and flammable organics that spill onto cell floors. Model development techniques and the ease of making model changes within the Mathcad environment are discussed. The results indicate that certain fluid spills result in overheating of the fluid, but the times to reach steady-state are several hundred hours. The long times allow time for spill clean up without the use of expensive mitigation controls.

Introduction

The Salt Disposition Integration Project (SDIP) at the Savannah River Site will impact the Defense Waste Processing Facility (DWPF). Within the DWPF, the Salt Processing Cell (SPC), Chemical Processing Cell (CPC), and Low Point Pump Pit (LPPP) contain numerous tanks of different solutions. Postulated events can result in the spilling of solution onto the cell floors, the loss of cooling to the tanks containing these solutions, and a combination where some solution spills to the floor and other solution containing tanks remain intact. The issue with the solution heat up is that the volatile organics in the solution can flash into the space above the solution, resulting in a flammable atmosphere. This calculation addresses two of the concerns associated with spilling the solution onto the cell floors. One concern is the solution temperature. If the solution temperature is high, organics within the solution will flash into the air space above the solution resulting in a potential flammability hazard. The second concern is the time to reach the steady-state temperature. If the times are long, then the facility will have time to clean up the spill, eliminating the flammability issue.

Some of the solutions that can spill onto the cell floor contain Cs-137 as the source of heat generation. Other solutions contain Cs-137 and other radionuclides. If sufficient heat is generated and the heat transfer rate from the solution is low enough, the solution can heat up to temperatures that result in safety concerns. This analysis determines the steady-state heat transfer conditions for the cells for various solution conditions in and outside the tanks, the steady-state temperature of the solution that are spilled onto the cell floors, and provides a conservative estimate of the time required to reach the steady-state conditions.

The predicted temperatures for various scenarios showed that the solution could overheat. Various design changes were evaluated to determine the impact on the solution temperature. The basic heat transfer model assumes no active systems are available for heat removal. Variations to the basic model were implemented to evaluate the impact on solution temperature of an air flow through the cell, the use of a purge flow into the vapor space above solution in a tank for cooling, and the use of a sparger for cooling. Sparger cooling is where a large nitrogen flow is inserted into the solution, allowing the nitrogen to heat up and provide some cooling of the solution.

Due to differences in building layout, the heat transfer model for the LPPP is different than the heat transfer model for the SPC and CPC. The model for the SPC and CPC is identical. The differences between the SPC and CPC are the input parameters to the model such as cell size and liquid spill contents. As the project evolved, variations in the original heat transfer models were developed to model building air flow, purge flow through the vapor space of a tank, and sparger flow through the liquid inside a tank.

Specific evaluations are completed for the SPC, CPC, and LPPP. An evaluation is completed for a spill from the Precipitate Reactor Feed Tank (PRFT) tank in the SPC. Within the CPC, spills can occur from the Strip Effluent Feed Tank (SEFT), Slurry Mix Evaporator Tank (SMET), Slurry Mix Evaporator Condensate Tank (SMECT), Sludge Receipt and Adjustment Tank (SRAT), Decontamination Wash Treatment Tank (DWTT), Melter Feed Tank (MFT), and Recycle Collection Tank (RCT). Within the LPPP, spills can occur from the Precipitate Pump Tank (PPT), Recycle Pump Tank (RPT), and Sludge Pump Tank (SPT). Within the LPPP, these tanks are located within their own cells. In addition to the spills, evaluations are completed for scenarios where the solutions are in the various tanks.

The solutions of interest for the SPC, CPC, and LPPP are Strip Effluent (SE) and SRAT Stream 7. The SE has a high Cs-137 loading while the SRAT Stream 7 solution has a combination of high Cs-137 and radionuclides. The solution temperatures are determined[1] for a range of solution activities, solution spill sizes, and environmental conditions.

ANALYTICAL METHODS AND COMPUTATIONS

The heat transfer models are discussed in this section. The generic building blocks that are used to create a heat transfer model are discussed followed by a general discussion for each model. The model for the SPC and CPC cell differs from the LPPP model. The model details are completed using the Mathcad software, and are presented provided in detail in Reference 1. The features for the different models are discussed in this section whereas the equation details are in Reference 1.

Material Properties

A full list of the material properties that are used is in Reference 1. For the steady-state calculation, only thermal conductivity property values are needed for the solids such as concrete, steel, and soil. For air, additional thermal properties such as dynamic viscosity, specific heat, and density are needed by the correlations for determining the heat transfer coefficients.
The thermal conductivity of the waste solution is assumed to be the value for water.

An estimate for the time to reach the steady-state conditions is also provided. This calculation requires the density and specific heat properties for concrete and water. The conservative calculation for time does not consider the impact of steel components in the cells, or the thermal mass of the soil that is considered in the calculation.

Components Used to Create a Heat Transfer Model

The heat transfer model is similar to a piping network model that considers fluid movement through pipes and nodes. By comparison, the heat transfer model considers heat movement through paths and nodes. In the heat transfer model, a path can be considered as heat transfer from one location to another location. Examples of a path are heat transfer from room air to a wall surface, heat transfer through the wall, and heat transfer off the back side of the wall. A node in the heat transfer model is where the heat flow has a choice of paths. An example of a node is the air in a room where heat transfer can occur to any of the vertical walls, the floor, or the roof. If air flow exists through the room, then heat transfer can occur to this air flow from the room node. Any type of air flow or water flow is modeled as a sink for heat transfer.

The heat flow in a path has resistance to the heat flow. For the piping network, the potential for flow is the pressure difference across the path whereas the potential for heat transfer is the temperature difference. The resistance in the piping network is a friction or form loss coefficient whereas the resistance to heat flow is thermal conductivity for conduction and a convection heat transfer coefficient for convective heat transfer. The resistance for radiation heat transfer is absorptivity and emissivity properties of the material along with the geometric shape factor.
In general, a convective heat transfer is modeled by.

$$Q = h * A * (T_h - T_c) \tag{1}$$

where Q = heat transfer, W
h = heat transfer coefficient, W / (m^2 – ^{o}C)
A = area for heat transfer, m^2
T_h = hot temperature, ^{o}C
T_c = cold temperature, ^{o}C

In Equation 1, the hot and cold temperatures represent two points. If natural convection between two surfaces is assumed, the hot and cold temperatures represent the two surfaces involved in the heat transfer. In this work if two surfaces are not involved, then the temperatures represent a surface and the adjacent air temperature. Equation 1 is valid for any convective heat transfer situation. The uniqueness of the geometry is represented by the heat transfer coefficient. Different correlations are used for different geometries and are discussed[1].

Conduction heat transfer is modeled by an equation of the form for flat surfaces.

$$Q = \frac{T_h - T_c}{\frac{1}{A} * \sum_n \frac{\Delta x_n}{k_n}} \tag{2}$$

where Q = heat transfer, W
Δx_n = thickness of the "n" material, m
k_n = thermal conductivity of the "n" material, W / (m – ^oC)
A = area for heat transfer, m^2
T_h = hot temperature, ^oC
T_c = cold temperature, ^oC

For cylindrical geometries, an equivalent equation to Equation 2 is used. In Equation 2, the value of "n" is determined by the number of materials. For conduction through the floor of the CPC or SPC, Equation 2 is used with n = 3 where the layers represent a steel liner, a concrete floor thickness, and a soil thickness.

Radiation heat transfer is used in this document for heat transfer off surfaces that experience a solar flux and is modeled by.

$$Q = A * \sigma * \varepsilon * (T_h^4 - T_c^4) \tag{3}$$

where Q = heat transfer, W
σ = Stefan-Boltzmann constant, W / (m^2 – $^oK^4$)
ε = emissivity for the material, unitless
A = area for heat transfer, m^2
T_h = hot temperature, ^oK
T_c = cold temperature, ^oK

The emissivity is a material dependent property. In this calculation, the hot temperature always represents a surface temperature of a material that receives a solar flux. In this study, Equation 3 is used for the roof surfaces of the CPC, SPC, and LPPP. Also, the south-facing walls of the LPPP building are modeled as receiving solar insolation and consequently use a radiative heat transfer mode defined by Equation 3.

Conservation of heat is accomplished at a node, similar to the conservation of mass at a node in a piping network. The heat into the node must equal the heat leaving the node.

For the heat transfer models in this document, the solar insolation onto a surface was multiplied by the absorptivity of the surface material to properly reflect the heat that the surface absorbed. The heat transfer models are composed of combinations of Equations 2 – 4. The unknown variables in the equations are heat transfer variables, heat fluxes, and temperatures. The number of unknown variables in a system, x, requires "x" equations and these equations are solved by the Mathcad software. For this work, the heat transfer models had as many as 67 equations to solve for 67 unknown variables.

The detailed listings of equations for each model are in Appendix B. In the Mathcad solution, the unknown variables are redefined as elements in a single column matrix, “s”. The Mathcad solution returns the solution for the variables in the single column matrix, “S”. The variables are redefined in terms of their solution values from the “S” matrix. The use of the single column matrix enables model modifications to be easily made. To increase model complexity with five additional equations requires the definition of an additional five components of the “s” matrix, and a redefinition of the five solution values from the “S” matrix.

The Mathcad software is used for this study. A full listing of each SPC, CPC, and LPPP model is provided[1].

Table 1 shows two techniques for solving equations within the Mathcad software. The native variable technique is recommended for a modest number of equations and unknown variables while the matrix notation is recommended for many equations and unknown variables. The drawback to the native variable technique is that for many unknown variables, the last statement with “Find” becomes very long. For this work with over 60 unknown variables, the “Find” statement will not fit on a single page. As Mathcad is very poor at line wrapping, the documentation is a problem. The matrix notation technique shown in Table 1 is very compact and amenable to a large equation set. To increase the number of equations, it is only necessary to define the new “s” variables, and insert the additional equations in the “Given-Find” block. After the “Find” statement, it is necessary to redefine the native variables in terms of the solution matrix “S”.

Table 1. Mathcad Solution Techniques.

Native Variables	**Matrix Notation**
T_h and Q are unknown variables.	T_h and Q are unknown variables.
Given $Q = \dfrac{T_h - T_{soil}}{\frac{1}{A}\cdot\left(\frac{\Delta x_1}{k_1} + \frac{\Delta x_2}{k_2}\right)}$ $Q = h\cdot A\cdot\left(T_h - T_{amb}\right)$ $\begin{pmatrix} T_h \\ Q \end{pmatrix} = Find(T_h, Q)$	$s_0 := T_h \qquad s_1 := Q$ Given $s_1 = \dfrac{s_0 - T_{soil}}{\frac{1}{A}\cdot\left(\frac{\Delta x_1}{k_1} + \frac{\Delta x_2}{k_2}\right)}$ $s_1 = h\cdot A\cdot\left(s_0 - T_{amb}\right)$ $S := \mathrm{Find}(s)$

Heat Generation in the Waste Streams

Three waste streams are evaluated in this work. One solution contains SE, a second solution is referred to as SRAT Stream 7, and the third solution is Sludge Stream 1. The heat generation in the SE is determined from a combination of the Cs-137 and Ba-137m radionuclides. Through β decay, the Cs-137 produces Ba-137m. The total heat generation in the SE is a combination of the Cs-137 and Ba-137m, and is calculated by the following formula[2].

$$q = q_{Cs\text{-}137} * (C_{Cs\text{-}137} + 0.946 * C_{Ba\text{-}137m}) \tag{5}$$

where q = total internal heat generation (W/gal)
$q_{Cs\text{-}137}$ = internal heat generation due to the Cs-137 (Ci/gal)
$C_{Cs\text{-}137}$ = conversion from Ci to watts for Cs-137 (W/Ci of Cs-137)
$C_{Ba\text{-}137m}$ = conversion from Ci to watts for Ba-137m (W/Ci of Ba-137m)

Equation 5 provides a conservative value for the heat generation because the Ba-137m contribution is due to gamma decay. Gamma energy is high and it is likely that a portion of this energy will not be deposited in the waste solution.

The SRAT Stream 7 solution contains sludge which has a Cs-137 component. The decay heat in the SRAT Stream 7 solution is a two-part calculation. One part considers all the radionuclides except for Cs-137 and the daughter product Ba-137m. The second part considers the Cs-137 and Ba-137m components. The calculation is divided into two parts because the SRAT Stream 7 for the calculations varied the Cs-137/Ba-137m content while keeping the remaining radionuclides constant.

The third solution is Sludge Stream 1 and contains a different combination of radionuclides than SRAT Stream 7. For this solution, the Cs-137 content is not varied for the calculations in this document.

Heat Transfer Model for the SPC

A schematic of the heat transfer model for the SPC is shown in Figure 1. There are multiple paths for removal of the heat from the SE fluid as shown in the figure. This model can consider a spill onto the cell floor, solution within the PRFT, a combination of solution on the floor and in the PRFT, air flow through the cell, and a sparger within the PRFT.

For the heat transfer model, passive heat removal from liquid spilled on the floor occurs by two paths. One heat transfer path occurs through the floor of the lower cell via conduction heat transfer into the ground. The conduction model assumes a steel liner, a concrete floor, and soil prior to reaching the steady-state soil temperature. The second heat transfer path occurs by natural convection to the air in the lower cell and then to the walls and cell cover of the lower cell. Heat transfer from the PRFT occurs to the air in the lower cell. Heat transfer from the cell air is modeled through the side walls into the portion of the DWPF building outside the cell. Additionally, heat transfer is modeled through the cell cover into the air space above the cell cover, and subsequently through the side walls and roof surrounding this upper cell air space. For the heat transfer through the vertical cell walls, the outside surface of these walls, which is the inside of the DWPF building, is considered to be at ambient conditions. If air flow into the lower cell is assumed, the air is assumed to act as a heat sink for the heat transferred into the lower cell.

At the outside building surface for the roof, the heat transfer considers solar insolation, radiative heat transfer off the building roof, natural convection off the building roof, and heat transfer by conduction from the lower surface of the roof to the upper surface of the roof.

Heat Transfer Model for the CPC

The heat transfer model is very similar to the SPC model in Figure 1. The differences are that the single PRFT tank in the SPC is replaced with seven tanks in the CPC. Each of the seven tanks is individually modeled as the various event scenarios assumed that the tanks contained different volumes of different waste streams. The schematic for the CPC is similar to the SPC model that is shown in Figure 1 and is provided in Reference 1.

Heat Transfer Model for the LPPP

The LPPP heat transfer model allows passive heat removal from fluid in the cell. The fluid can be on the cell floor, in the tank within the cell, or a combination of solution on the floor and in the tank. As for the SPC and CPC model, one heat transfer path occurs through the floor of the cell into the ground by conduction heat transfer. The heat in the air space of the various cells in the LPPP is transferred to the vertical walls that are exposed to soil and through the removable concrete roof. For the LPPP, there are two variations of the heat transfer model. One variation models three vertical exterior walls for the PPT and SPT cells while the model variation for the RPT cell considers two vertical exterior walls. The PPT and SPT cells are located at the north and south ends of the LPPP building while the RPT cell is between the PPT and SPT cells. The cells are constructed such that their roof sits at ground level. Each model variation allows heat transfer to the ambient surroundings through the LPPP building.

The LPPP building covers the PPT, RPT, and SPT cells from the surroundings. The LPPP building is modeled as two rectangular structures. One large structure is over the cells while a smaller structure is connected to the larger structure. The connecting wall is a gypsum over steel frame design. The composite thermal conductivity for the gypsum wall is maximized to provide the greatest heat transfer to the building air above the LPPP cells. Similar to the heat transfer off the roofs of the SPC and CPC, the heat transfer model only considers a solar insolation to impact the roofs and south-facing walls of the small and large sections of the LPPP building. Consequently, these surfaces are the only outside building surfaces to use the radiative heat transfer model.

The LPPP models are very sensitive to the thickness of the soil that is adjacent to the vertical walls and floor. A soil thickness is needed to provide a transition from the warm concrete surface to the point in the soil where the year-round temperature is reached. The thickness chosen for the model is based on a heat transfer study from the Saltstone vault[3]. The value for the soil and concrete thickness is six meters.

Reference 1 provides a schematic of the heat transfer model for the LPPP models.

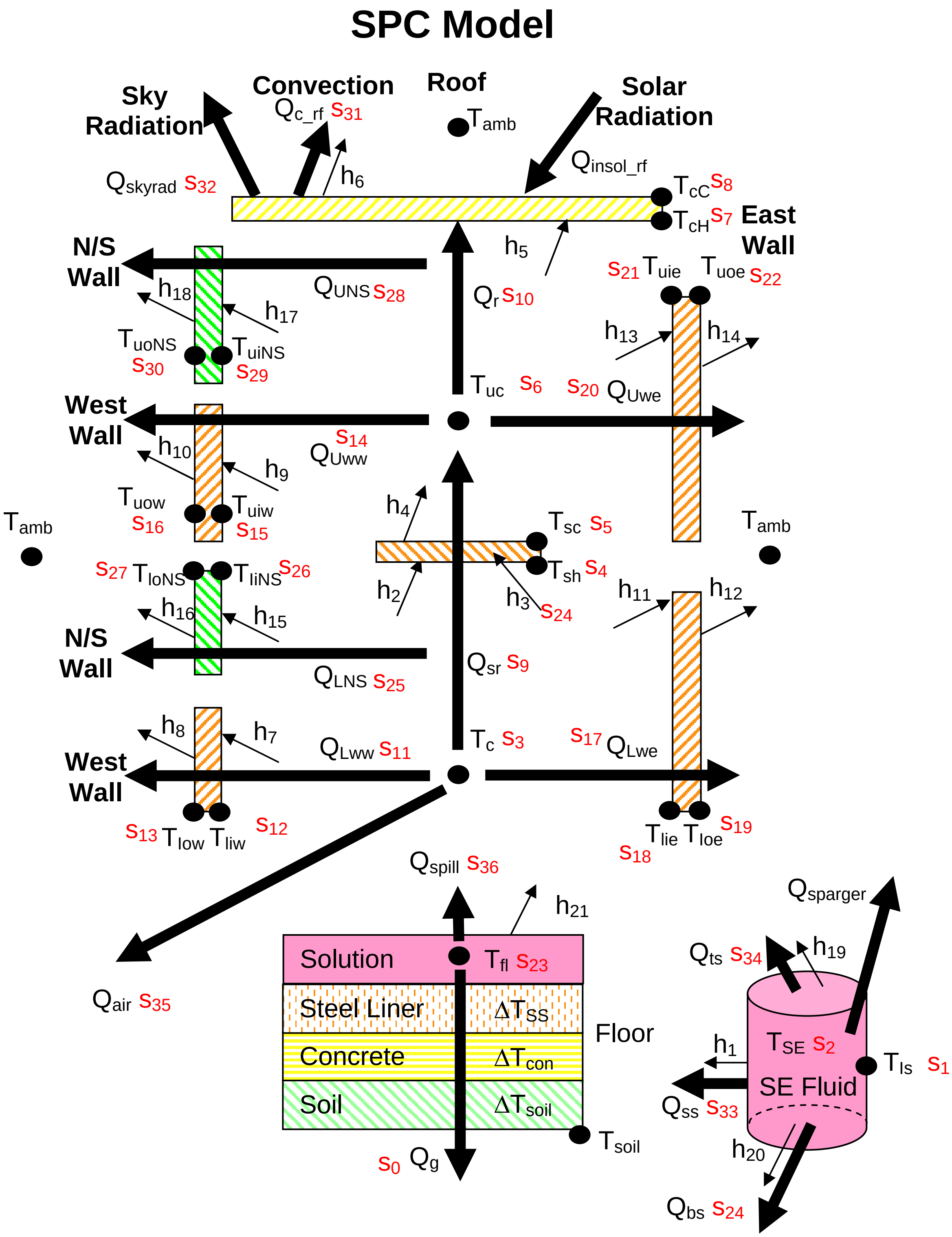

Figure 1 – Heat Transfer Model for the SPC.

Time to Reach Steady-State Conditions

The heat transfer model determines the steady-state solution. A conservative estimate of the time to reach the steady-state condition is determined by calculating the heat necessary to heat up the concrete floor, roof, and walls, and waste solution from initial temperatures to the final steady-state conditions. For this calculation, the starting temperature of the component is assumed. An average temperature of the component at the steady-state condition is calculated. The heat transfer model assumes that the waste solution is at an average temperature.

The general equation for determining the time for steady-state to be reached is.

$$Q \cdot time = \sum_{n=1}^{N} \left[\rho \cdot V \cdot Cp \cdot \left(T_{end} - T_{start} \right) \right] \qquad (6)$$

where Q = heat generation in the SE fluid, W
time = time to heat up from T_{start} to T_{max}, hrs
ρ = density of n^{th} component, kg/m^3
V_{spill_tot} = volume of the n^{th} component, m^3
Cp = specific heat of the n^{th} component, J / (kg - ^oC)
T_{max} = end temperature of the n^{th} component for the heat up, ^oC
T_{start} = starting temperature of the n^{th} component, ^oC
N = number of components

Results

The results should be considered tentative as the work is ongoing. The final results will be documented[1] once the work is completed.

The heat transfer models for the SPC and CPC are expected to show the same sensitivity to inputs and modeling techniques as the only differences between the cells are the basic cell dimensions. The heat transfer model for the SPC was evaluated for sensitivity to variations in input parameters and to variations in model parameters. Table 2 provides a summary of these results. The base information for the Table 2 results is a SE fluid spill of 9,000 gallons at 66 Ci/gal of Cs-137 onto the SPC floor.

In evaluating the results of the heat transfer model, the following items should be considered. For the SPC and CPC, heat transfer occurs across 18 surfaces. Small variations in heat transfer coefficient on one surface have negligible impact on the SE fluid temperature. Conditions on the roof of the building also have minimal impact on the SE fluid temperature due to the heat transfer characteristics of the steel cell cover.

Table 3 presents the SPC results for different volumes of SE fluid. The 3,375 gallons is the nominal spill volume for SE fluid. The 7,200 gallon volume is the operational volume for the PRFT tank while the maximum volume of the PRFT tank is 9,000 gallons.

Table 2. Model Sensitivity to Variations in Input and Model Parameters.

Input Parameter Variation from Nominal	SE Fluid Temp, ^{o}C
Nominal case.	41.9
Soil thickness increased by a factor of 2	43.0
Ambient temperature increased by 2.8^{o}C to 30.3^{o}C	44.5
Solar insolation decreased from 7 to 4.6 kW-hr / m^{2} / day	41.9
Length scales used in heat transfer correlations increased by a factor of 2	41.9
Convective heat transfer coefficients decreased by 20%	44.6
Convective heat transfer coefficients increased by 20%	40.0
Effective sky temperature for radiative heat transfer decreased from ambient to 9.5^{o}C	41.7
Thermal conductivity of concrete increased 38%	41.2
Notes: 1. Nominal case uses an average July insolation value of 7 4.6 kW-hr / m^{2} / day. 2. The ambient temperature condition is 27.6^{o}C. 3. Nominal concrete plus soil thickness is six meters.	

Table 3. SE Fluid Temperatures for the SPC for Various Loadings.

SE Fluid Spill, gallons	Temperature, ^{o}C			Heat Up Time to Reach Steady State, hrs
	SE Fluid	Lower Cell Air	Inside Roof Surface	
3,375	33.4	30.2	30.7	NA
7,200	39.5	33.0	32.9	899
9,000	41.9	34.1	33.4	1120

Table 4 presents results for a combination case with a spill to the CPC floor as well as solution within tanks. For these cases, the SE fluid is assumed to be on the cell floor leaving the SEFT empty. The other tanks retain their solution. The heat transfer model does not consider boiling heat transfer; thus, the results for those tanks showing temperatures well above 100^{o}C are unrealistically high. Table 4 shows the significant impact of air flow through the cell.

Table 4. Results for Floor Spills and Solution in Tanks for the CPC.

CPC Configuration	Fluid Temperature, ^{o}C	
	No Air Flow	10,000 cfm Air Flow
12,000 gal 125 Ci/gal SRAT Stream 7 in SRAT	112.0	73.4
500 gal 250 Ci/gal SRAT Stream 7 in RCT	74.0	37.5
0 gal in SEFT	68.8	32.6
12,000 gal 125 Ci/gal SRAT Stream 7 in SME	112.0	73.0
2,000 gal 250 Ci/gal SRAT Stream 7 in SMECT	83.6	46.5
12,000 gal 250 Ci/gal SRAT Stream 7 in MFT	127.4	87.9
500 gal 250 Ci/gal SRAT Stream 7 in DWTT	74.0	37.5
21,000 gal 66 Ci/gal SE on cell floor	72.4	38.0
Time to reach steady-state, hrs	1055	457

Table 5 shows the solution temperatures for the cases where some solution spills to the cell floor and some solution remains inside the tanks in the LPPP. The temperatures are higher in the LPPP than in either the SPC or CPC because the LPPP is underground, and the ability of the soil to remove heat is poor.

Table 5. Results for the SE Fluid on the Floor and in the Tank in a LPPP Cell

Event	Temperatures, ^{o}C	
	PPT and SPT Cells	RPT Cell
2000 gallon tank 2000 gallon spill	52.9 54.5	57.5 58.9
2000 gallon tank 1500 gallon spill	50.4 50.8	54.7 54.8
2000 gallon tank 500 gallon spill	45.3 42.5	49.0 45.8
3000 gallon tank 500 gallon spill	52.0 47.4	56.4 50.8

Conclusions

Steady-state heat transfer models for the SPC, CPC, and LPPP cells were developed. Results show that maximum loadings from the process can result in high temperatures if all cooling to a cell is lost. The model predicts that peak waste stream temperatures can be reduced if only a modest air flow is maintained through a cell.

References

1. McAllister, J.E., “Evaluation of Heat Transfer from Heat Generating Liquids in the SPC, CPC, and LPPP (U)”, S-CLC-S-00126, Washington Safety Management Solutions, Aiken SC, To Be Issued.

2. Norato, M.A., “DWPF Material and Curie Balance for the solvent and Cs137/Ba137m as a Result of the Addition of the MCU Cesium Strip Solution”, X-CLC-S-00140, Rev. 2, Washington Savannah River Site, Aiken SC, January 2006.

3. Shadday, M.A., “Analysis of Thermal Overshoot in Saltstone pouring”, M-CLC-A-00293, Rev. 0, Washington Savannah River Site, Aiken SC, December 2006.