Paper # 5084 Canadian Nuclear Society
Waste Management, Decommissioning and Environmental Restoration
for Canada's Nuclear Activities: Current Practices and Future Needs
Ottawa, Ontario Canada May 8-11 2005

Assessing the Performance of Passive Ventilation for a Used Fuel Dry Storage Facility

Kevin Matsui P. Eng. kevin.matsui@ansys.com
and
Judy Cooper judy.cooper@ansys.com
ANSYS Canada Ltd
519-886-8000
www.ansys.com
554 Parkside Drive
Waterloo Ontario Canada
N2L 5Z4

Marty Waldinger P. Eng. marty.waldinger@opg.com
Ontario Power Generation
Nuclear Waste Management Division
700 University Ave H16 B26
Toronto Ontario Canada
M5G 1X6

===

ABSTRACT

Dry storage is a proven technology in use around the world to store used, or spent nuclear fuel containers. The design of the storage facility must provide adequate passive ventilation to maintain the ambient air temperature in the space containing the DSCs at or below the specified design temperature.

A computational fluid dynamics (CFD) method was chosen to verify the ventilation design for the Western Used Fuel Dry Storage Facility (WUFDSF).

===

Introduction

Ontario Power Generation's Dry Storage Container (DSC) Storage Buildings use passive ventilation through wall and roof louvres to dissipate decay heat from used nuclear fuel in storage to the atmosphere. The system maintains the storage area below an average ambient temperature of 38°C in the proximity of DSCs.

A computational fluid dynamics (CFD) method was used to verify the ventilation design for the Western Used Fuel Dry Storage Facility (WUFDSF).

This paper details the results of the CFD analysis.

Background

Dry storage of used fuel, initially cooled in Irradiated Fuel Bays for several years after discharge from the reactor, is used worldwide as the preferred method for the safe and economical storage of used fuel.

The Western Used Fuel Dry Storage Facility (WUFDSF) is situated on the Nuclear Waste Management site in Bruce County, Ontario, Figure 1. The WUFDSF provides storage for used fuel generated by eight CANDU nuclear reactors on the Bruce Nuclear Power Development (BNPD) site. The DSC Storage Buildings use passive ventilation through wall and roof louvres to dissipate decay heat from used fuel in storage to the atmosphere to maintain the storage area below an average ambient temperature of 38°C in the proximity of DSCs.

The ambient temperature of 38°C is based on heat transfer to maintain the maximum sheath temperature for ten-year cooled Bruce used fuel to be less than 175° in dry storage in a helium atmosphere. At these low temperatures, only the volatile fission products would be released should the fuel sheath become damaged.

The design of the Storage building was based on American Society of Heating, Refrigeration and Air conditioning, Engineers (ASHRAE) principles to determine louvre sizes required to provide necessary cooling by passive ventilation. A computational fluid dynamics (CFD) method was chosen to verify the original ventilation design for the Western Used Fuel Dry Storage Facility (WUFDSF).

Development of a Mathematical Model of the Storage Building

CFD Modeling Approach

The geometry of the dry storage facility was relatively simple, as it consisted of a rectangular building with a 2 per cent sloped roof. The most crucial element of this study was the choice of models for dry storage containers (DSCs). The proposed facility contained 490 DSCs, arranged in 7 distinct grids as shown in Figure 2. Based on the design conditions, the largest clearance between the containers was 0.635 m and the smallest clearance was 0.230 m. The reduced ventilation conditions imposed by these small clearances necessitated a detailed study in which every DSC was individually represented in the model. The model was therefore created in which each DSC was treated as an inert solid with a surface heat flux equivalent to that of the actual containers. This choice of model was deemed to give the most conservative heating prediction, since it included potential stagnant “hot spots” between the containers.

Because of the non-uniform ventilation and insolation conditions experienced by the walls of the proposed facility, no symmetry condition presented itself. The CFD model therefore consisted of the entire building interior, as shown in Figure 2.

The geometry was described in Ontario Power Generation dimensioned drawings. A solid model of the facility was created. The surfaces of the 3-dimensional solid were then extracted into the ANSYS ICEM CFD Hexa mesh generation software and used to define the bounding faces of the volume mesh. A hexahedral mesh was produced for computational efficiency.

The mesh spacing was specified so that there was at least one solution node in the air gaps between the DSCs. This produced enough grid resolution to capture the heat lost from the DSC surfaces. The resulting volume mesh had 875,000 nodes. Aspect ratios for the mesh were between 1.0 and 36.4.

CFX solves the three-dimensional Reynolds-averaged Navier-Stokes equations using a collocated, finite volume, fundamental variables (Cartesian components of velocity) approach [R-01, R-02, R-03]. It is applicable to flows ranging from fully incompressible to transonic to supersonic. The flow code is a finite element based finite volume method in the sense that a finite element assembly procedure is used to assemble conservative finite volume equations. The equations are discretized using a scheme known as Linear Profile Skewed upwind differencing (LPS) with Physical Advection Correction [R-05, R-06]. This discretization scheme is second order accurate, and has very low levels of false diffusion with excellent conservation of stagnation quantities (total pressure and temperature). CFX computes dynamic pressure (thermodynamic less hydrostatic). Turbulence is modeled with the standard two-equation k-e model.

The discrete linearized algebraic equations are solved using a powerful multigrid method known as Additive Correction Multigrid [R-05, R-06]. Due to this multigrid method, solution costs are observed to increase only linearly with increase in grid size.

Physical Properties and Boundary Conditions

The properties of the ambient air were allowed to vary with temperature according to the ideal gas law. The boundary conditions used in the simulation consisted of wall boundaries and opening boundaries. The orientation of the building with respect to the application of the various boundary conditions is shown in Figure 3.

The boundary conditions were applied as follows:

Wall Boundary: The closed bounding faces of the flow domain.

- All walls were considered to be hydrodynamically smooth.
- General wall boundary conditions were adiabatic, neglecting the effects of heat conduction through the wall and convective cooling due to winds.

ROOF: A fixed heat flux of 50 W/m^2 was applied to the roof to take into account the average daytime insolation. This flux was calculated by averaging a sinusoidal relationship for the daily estimated insolation and then taking ten percent of this to account for the effects of the building insulation. Night time heat losses were omitted from all averaged insolation to compensate for any overestimation of building insulation functionality. An additional 7 W/m^2 was applied evenly over the roof area to account for the 36.8 kW of heat generated by the lighting system. The total heat load to the roof (insolation and lighting) was 300 kW.

WALLS: A heat flux of 12.5 W/m^2 was applied to the west wall to account for daily insolation on this side of the building. This value was calculated in same manner as the roof heat flux. The area of this wall estimated to conduct heat into the building was 680 m^2. The total heat load from this wall was 8.5 kW.

TRANSPORTER: An additional heat flux of 95 W/m^2 was applied over the floor in the main aisle of the building to account for the heat release (approximately 73 kW) generated by the DSC transporter. This aisle runs north south in the building. The heat generated by the transporter was estimated based on the transporter being present 10% of the time.

DSCs: Characterization of the internal volume of the DSCs was omitted from the model. An equivalent surface heat flux was therefore required to simulate the heat produced by the fuel inside them. The total heat produced by the fuel in the DSCs was calculated on the basis of the average heat generated by a fuel bundle over the 5-year period taken to fully stock the facility, or 5.5 W. Each DSC contained 384 fuel bundles. The facility was considered to contain 500 DSCs at full capacity, which was 8 DSCs in excess of what would realistically be stored. The total heat attributed to the DSCs was therefore 500 x 384 bundles x 5.5 W/bundle, or 1056 kW. The estimated total surface area of the DSCs was 490 x 37.4 m^2, or 18326 m^2. An equivalent heat flux of 57.6 W/m^2 was applied over the exterior surfaces of the DSCs to represent the heat generated by the fuel.

Opening Boundary: All louvers were defined using an opening boundary with the direction of flow implicit in the flow solution. This was done to provide information about the natural direction of the buoyantly driven flow and facilitate an optimization of the final louver slat orientations. Because the slats of the louvers were not included in the analysis, the height of all louver openings was reduced to 41.5% of the original value to simulate the obstruction produced by the slats. The horizontal centerline for both wall and roof louvers were maintained at their original elevations. This effective area reduction was an average of the 40.7% open area specified for the wall louvers and the 42.3% specified for the roof louvers.

A reference pressure of 1 atmosphere (101.3 kPa) and a reference temperature of 25 °C were specified as ambient conditions.

Based on the estimated flow rate, the Reynolds number for the flow along the flow path was determined to be of the order of $8x10^5$. Therefore the flow was modeled as turbulent and the turbulence was treated by the standard k - e model with standard wall functions. A reference temperature of 298 K and gravity force of -9.81 m/s^2 were specified for the flow domain in order to drive the buoyant flow.

Results

The results of the computation of the flow field are demonstrated in a series of temperature fringe plots, vector plots, and isosurface plots of temperature. Temperature and velocity plots are presented at several cross sections to give a better impression of the overall flow in the facility. Figures 4 to 7 show typical plots.

Buoyant flows are usually fairly unsteady, but, because of the relatively weak buoyant forces involved in this case, the CFD solution was very well behaved. This fact indicates that the actual dry storage facility will attain a well-defined steady state and experience few fluctuations once this state has been achieved. The results of the predicted flow field show that the design of the proposed facility will provide enough airflow to keep temperatures around the DSCs at desirable levels.

The solution showed that the placement of the roof louvers along the eastern side of the building produced an east-west temperature gradient in the building that was noticeable as far down as 5 m from the floor of the building. Building temperatures were hottest in the Southwest corner of the building because of the insolation heat load on the west wall and the presence of the processing building, which obscured part of the louver line on the south end of the building. The Northeast corner of the building was the coolest because of the unobscured louver run and the fact that no insolation heat flux was applied to the east wall. This temperature trend would suggest that, for greatest cooling efficiency, the facility could be stocked from Southwest to Northeast, with the newest (hottest) fuel located in the areas of greatest cooling.

The ambient temperature in the centre of the gap between DSCs is estimated to be below 38°C. The solution showed that the temperatures between adjacent DSCs ranged from 25°C to 45°C (the higher temperatures are in the air layer immediately in contact with the DSC). The solution showed that the maximum temperatures in the region of principal concern were most likely to occur between the DSCs. This was particularly true in the centre of each DSC array, because of the greatly reduced cooling airflow in these locations. The higher end of this temperature range likely stems from the representation of the DSC. Applying the representative heat flux for the DSCs over their surface, rather than through an internal volumetric heat source equivalent to the fuel bundles results in temperature over prediction at the DSC surface because the internal temperature gradient of the DSC is omitted. The model of the individual DSC assumes that the heat output is evenly distributed over the entire surface. The actual DSC has the heat source in a cavity within the container.

The characteristics of the building flow patterns are useful in determining the optimum orientation of the louver slats. The flow pattern in the facility was such that flow was directed in toward the centre of the building through the wall louvers and out through the roof louvers. The rate of air throughout for the fully loaded facility was predicted to be on the order of 138 kg/s (122.8 m^3/s). A more detailed breakdown of the louver bulk flow rates is shown below in Table 2. The volume flow rates in the table calculated based on a bulk density of 1.125 kg/m^3. The louvers numbered 1-4 are wall louvers and 5-6 are roof louvers. Building outflows are denoted by negative quantities.

Table 1: Flow rates through building louvers

Louver	Mass Flow (kg/s)	Effective (Open) Area (m^2)	Volumetric Flow Rate (m^3/s)	Velocity (m/s)
L1 (SW)	8.8	5.9	7.8	1.3
L2 (W)	43.3	70.4	38.5	0.55
L2 (E)	42.9	70.4	38.1	0.54
L3 (N)	35.8	29.3	31.8	1.1
L4 (SE)	7.4	7.2	6.6	0.92
L5	-107.4	86.7	-95.5	-1.1
L6	-30.7	22.0	-27.3	-1.2

Conclusions

The designs of the Used Fuel Dry Storage buildings will provide more than adequate cooling for the Dry Storage Containers. For this application, the ASHRAE principles provided conservative results. The use of a computational fluid dynamics (CFD) method in the initial design phase could result in a design optimized for passive ventilation. This could reduce the required louver area and building cost.

References

[R-01] M. J. Raw, P. F. Galpin and B. R. Hutchinson, "A Colocated Finite Volume Method for Solving the Navier-Stokes Equations for Incompressible and Compressible Flows in Turbomachinery: Results and Applications", First Canadian Symposium on Aerospace Propulsion, Ottawa, Ontario, Canada, May 15 and 16, 1989.

[R-02] B.R. Hutchinson and G.D. Raithby, "A Multigrid Method Based on the Additive Correction Strategy", Numerical Heat Transfer, Vol. 9, pp. 511-537, 1986.

[R-03] B.R. Hutchinson, P.F. Galpin and G.D. Raithby, "Application of Additive Correction Multigrid to the Coupled Fluid Flow Equations", Numerical Heat Transfer}, Vol. 13, pp. 133-147, 1988.

[R-04] P.F. Galpin, R.G. Huget, G.D. Raithby, "Fluid Flow Simulations in Complex Geometries", CNS/ANS Conference on Simulation Methods in Nuclear Engineering, Montreal, Oct. 1986.

[R-05] J.P. Van Doormaal, A. Turan and G.D. Raithby, "Evaluation of New Techniques for the Calculation of Internal Recirculating Flows", AIAA paper No. 87-0059, 25th Aerospace Sciences Meeting, Reno, 1987.

Figure 1: Aerial view of Nuclear Waste Management site in Bruce County, Ontario

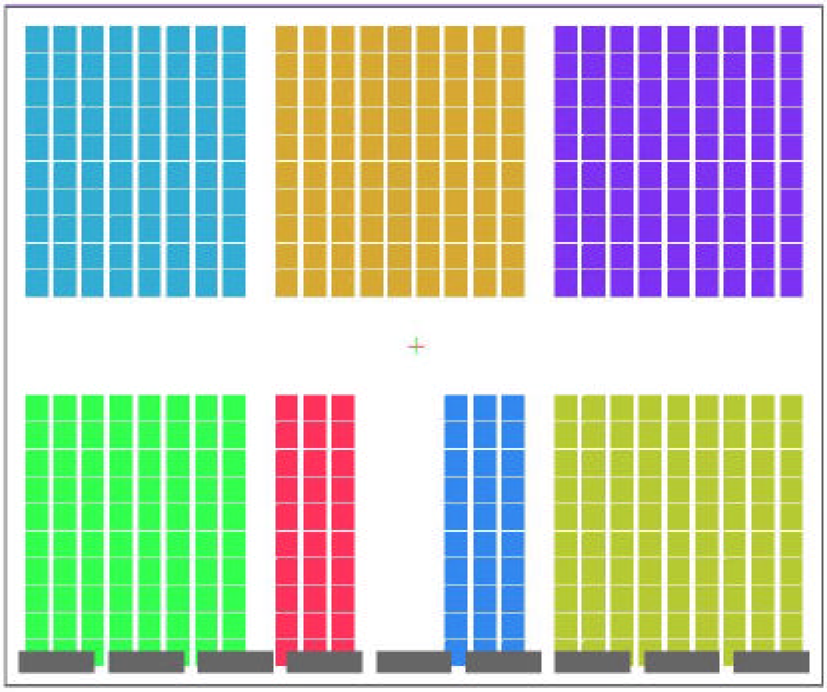

Figure 2: Storage Layout Containing 490 DSCs in 7 Grids

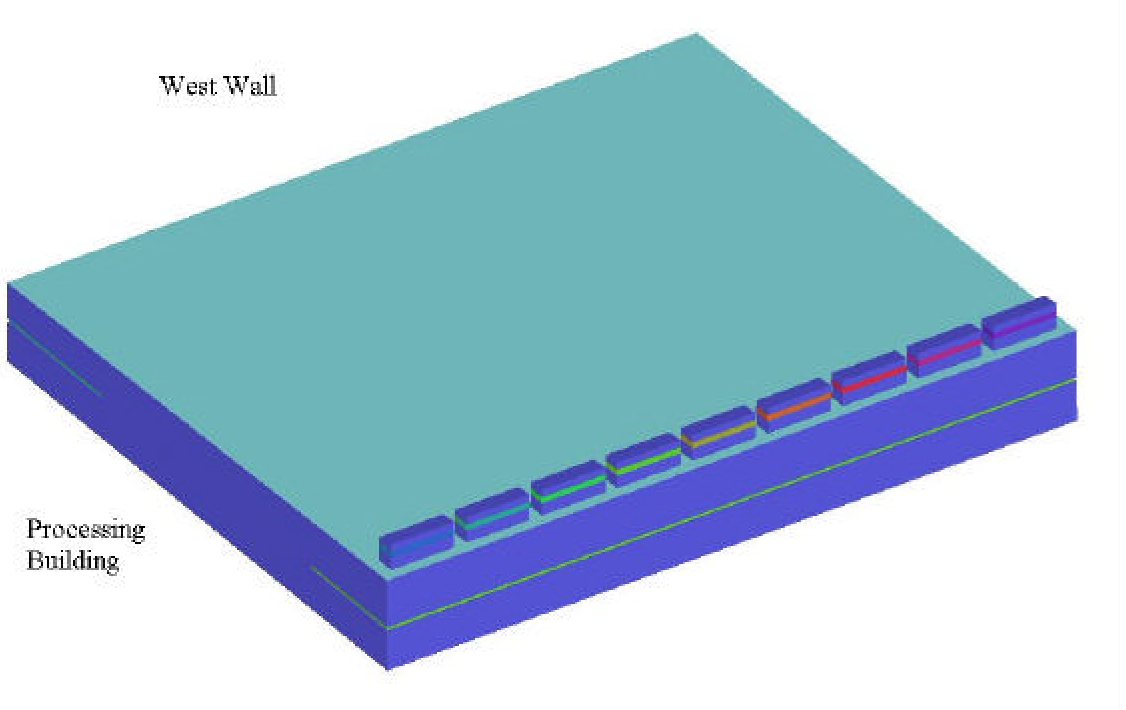

Figure 3: Isometric view of building exterior showing roof louvers and orientation (+z axis points upward)

Figure 4: Isosurface of 35 °C temperature showing shape of hot layer strata (temperatures greater than 35 °C lie above and inside yellow surfaces)

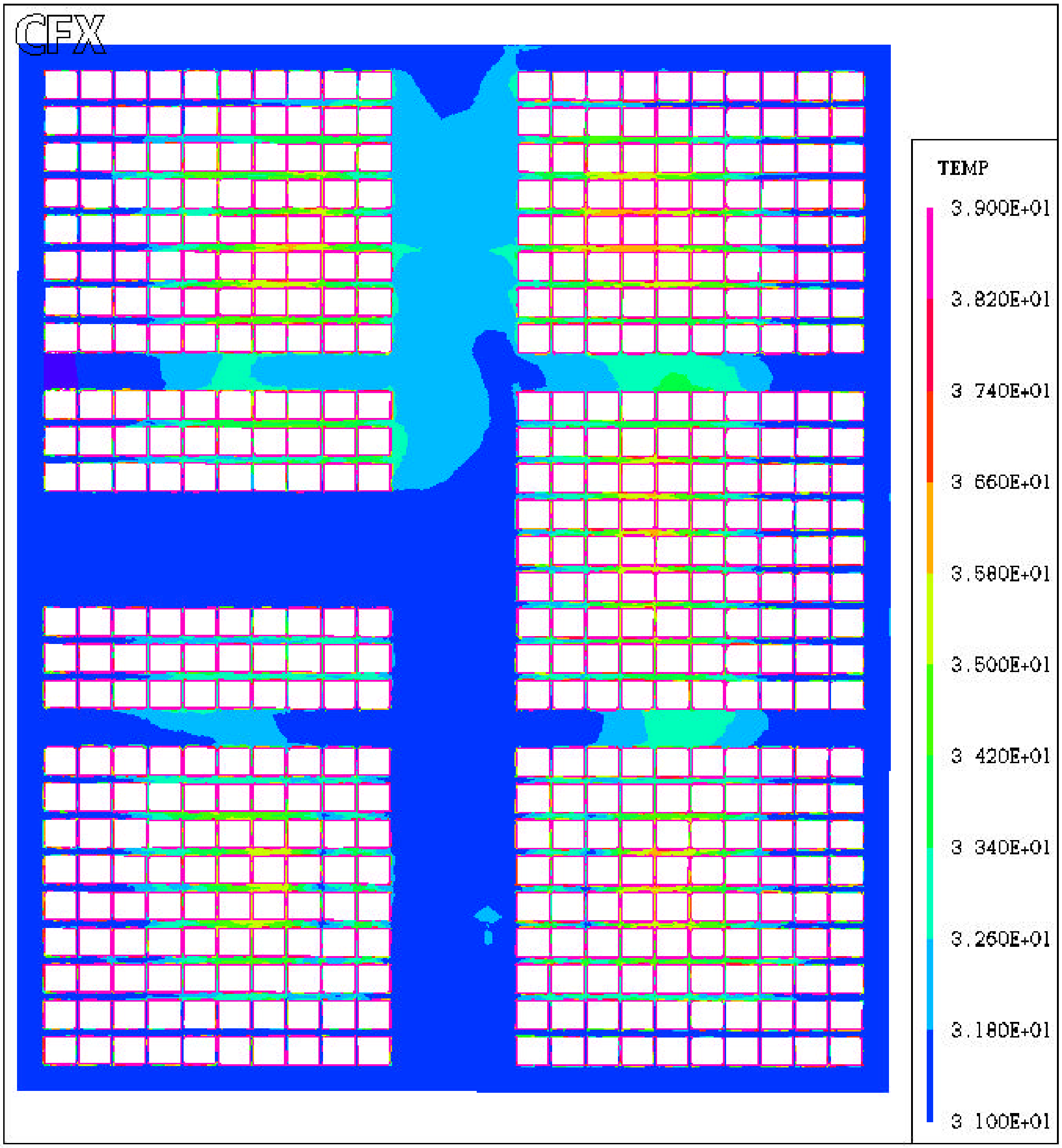

Figure 5: Temperature contours at 3 m elevation showing heating pattern between DSCs (Magenta areas over 38 ^{o}C)

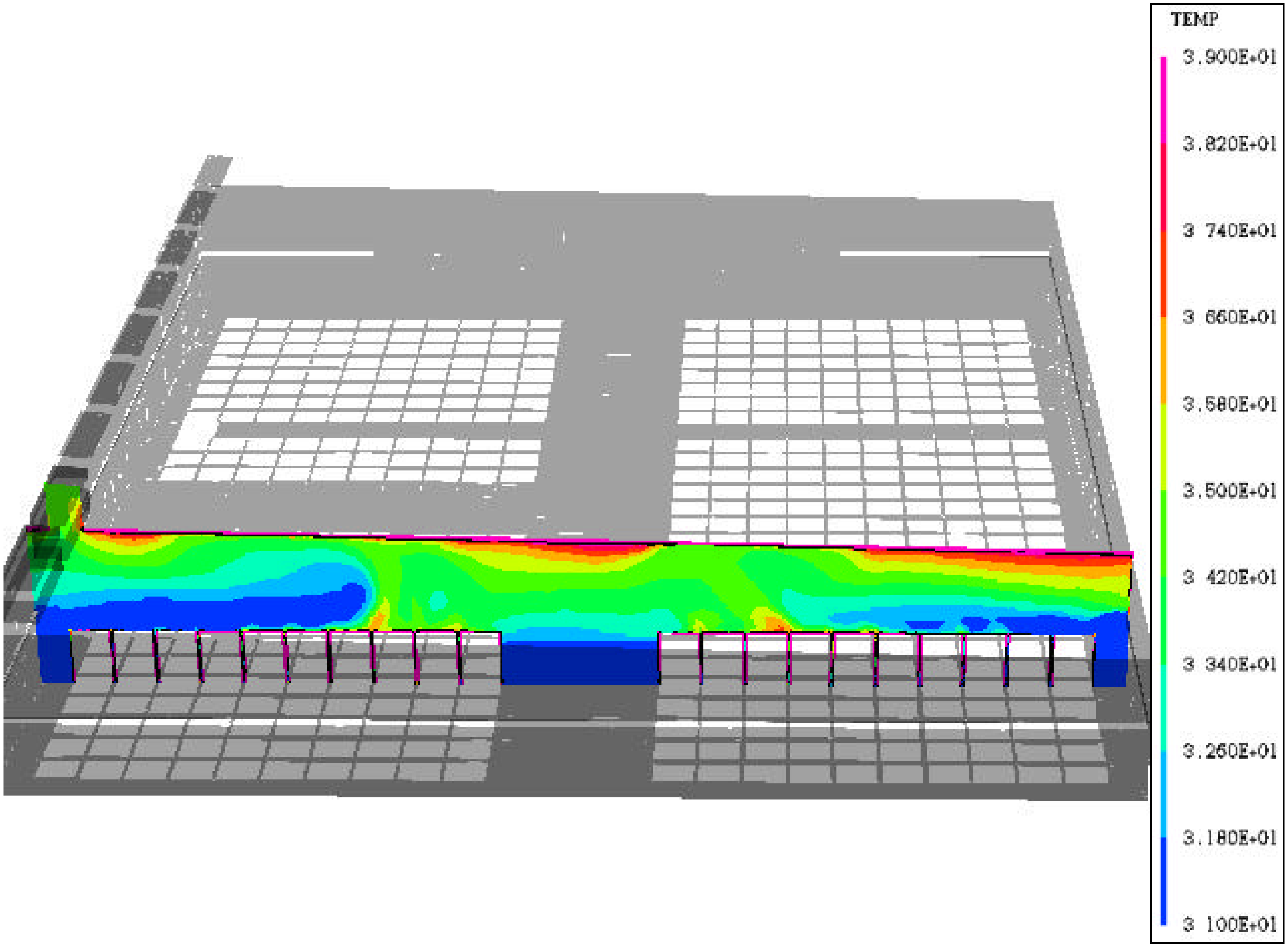

Figure 6: Temperature Contour on vertical plane of DSC grid

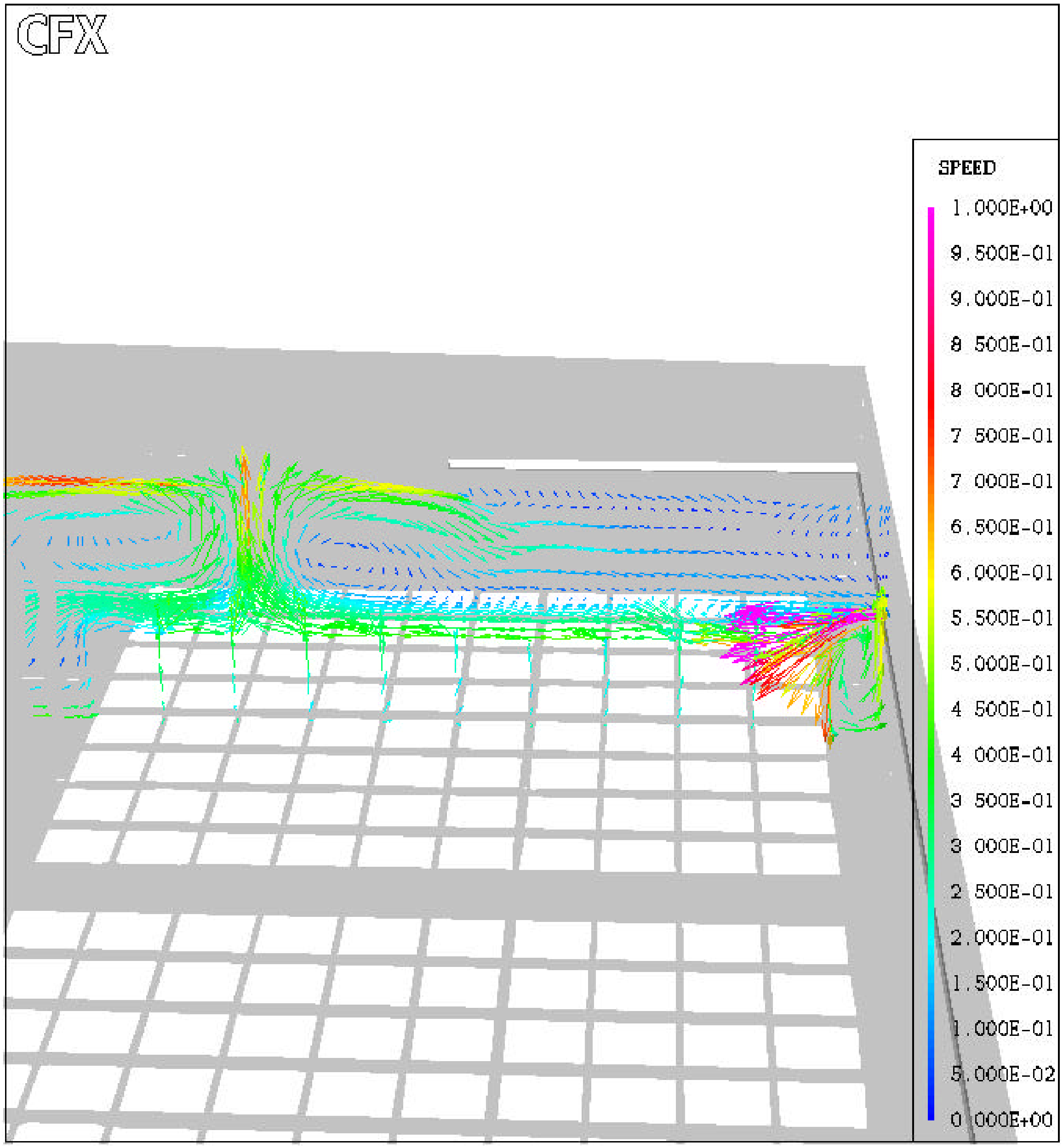

Figure 7: Vertical cross section detail of flow pattern near southeast corner of the facility (speed in m/s).