HEFAT2012
9th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
16 – 18 July 2012
Malta

CFD SIMULATIONS OF THE SPENT FUEL POOL IN THE LOSS OF COOLANT ACCIDENT

Lin Y.T., Chiu P.H. *, Chen B.Y., Chang C.J. and Chan Y.K.
*Author for correspondence
Nuclear Engineering Division,
Institute of Nuclear Energy Research,
Taoyuan County, Taiwan,
Republic of China,
E-mail: phchiu@iner.gov.tw

ABSTRACT

The study utilized the computational fluid dynamics (CFD) methodology to investigate the thermal hydraulic behavior during the hypothetical event of normal operation and loss of cooling accident occurring at spent fuel pool. The boiling time, water level decreasing rate, fuel exposure time and temperature response after fuel exposure for the nuclear power plants under the accident were predicted in this study. We also analyze the flow and heat transfer for the single Atrium-10 fuel bundle. The details of the physics will be shown in this study. The results indicate that the fuel temperature in the pool will not exceed 1200°C to avoid the water-metal reaction after failure of RHR system for 4.578 days. We find that the velocity in the bundle are much faster than outside of the bundle under the LOCA accident.

INTRODUCTION

A great earthquake could make the nuclear power plant emergency shutdown due to the fact that the earthquake intensity exceeds the design value of G force. The spent-fuel-pool could be damaged severely even when the main structures of the containments remain its integrity. When this scenario happens, the RHR system usually fail to maintain the designed temperature of the spent-fuel-pool. This makes the water-level decrease, spent-fuel exposure, spent-fuel's temperature increase and finally cause explode due to the water-metal-interaction. The LOCA (Loss of coolant accident) of spent fuel pool also causes in the severe structure broken problem of fuels and bundles. This is due to the water-metal interaction and melting of the zirconium shells by the decay heat. This motivates us to investigate the heat transfer characteristics of spent fuels in the Loss of coolant accident and estimate the allowable time for refilling the water in the spent-fuel-pool by using the CFD simulation-tool in this study.

The rest of this paper is organized as follows. The methodology for predicting the decreasing water-level of the spent-fuel-pool will be presented. It is followed by presenting the method used for predicting the decreasing water-level of the spent-fuel-pool. The Atrium-10 spent fuel bundle, including fuel rod, helium layer, zirconium alloys shell, rack structure and boron carbide will be completely modelled in this study. The CFD commercial software, ANSYS Fluent, is used to solve the flow, heat transfer and radiation equations. By analyzing cases of normal operation and LOCA accident of spent fuels, we also propose the allowable uncovered spent fuels length and the distributions of the temperature of the spent fuel bundles in these situations. Finally, we draw some conclusions in final section.

GOVERNING EQUATIONS AND TURBULENT MODEL

The coolant flow in the Atrium-10 [1] fuel bundle shown in Fig. 1 can be extremely complex. In pool environment, coolant is a phenomenon of great complexity due to its interaction with the decay heat and buoyancy effect.. Multi-scaled and rotational eddies carried along in a coolant stream moving relative to the bundle surface are typical characteristics and can result in increase of the coolant temperature.

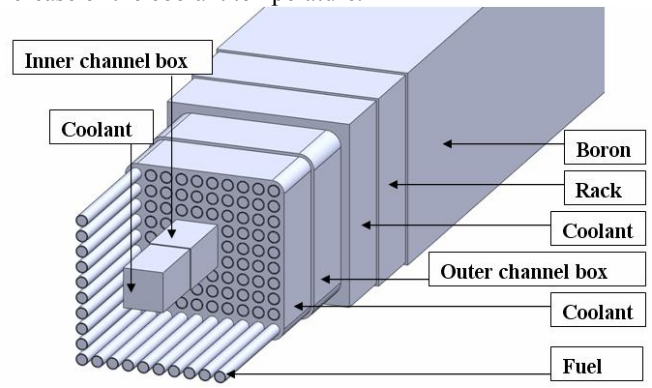

Figure 1 Schematic of the fuel bundle.

As a result, an accurate modeling of conjugate heat transfer and buoyancy effect of the spent-fuel bundle heat transfer characteristics must take the effect of turbulence into account. Turbulent details cannot be resolved numerically on the currently available computers for large-scale flow simulations. It is therefore the common practice to solve the Reynolds-averaged Navier-Stokes equations together with the turbulence models to compute the averaged turbulent stresses. Within the incompressible flow context, appropriate conservation equations for modeling the time-dependent viscous airflow around a group of buildings of complex shapes can be expressed as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \quad (1)$$

$$\frac{\partial \rho \bar{V}}{\partial t} + \nabla \cdot (\rho \vec{V} \vec{V}) = -\nabla p + \nabla \cdot (\mu_{eff}(\nabla \vec{V} + (\nabla \vec{V})^T) + \vec{g}) \quad (2)$$

$$\rho C_p (\frac{\partial T}{\partial t} + \nabla \cdot (\vec{V} T)) = k_T \nabla^2 T \quad (3)$$

where ρ, t, $\vec{V}$, p, T, μ_{eff} and k_T represent the fluid density, time, fluid velocity vector, static pressure temperature, fluid viscosity and thermal conductivity. Note that the above effective viscosity can be decomposed as the sum of the laminar viscosity μ_L and the turbulent viscosityμ_T, implying that $\mu_{eff} = \mu_L + \mu_T$.

The $k-\epsilon$ model has been considered as the most reliable ones with a reasonable accuracy and computational cost in predicting turbulence flows, we employ this model in the present study to calculate μ_T by means of the Boussinesq eddy viscosity assumption given by $\mu_T = C_\mu \rho \frac{k^2}{\epsilon}$, where k and ϵ represent the turbulent kinetic viscosity and the turbulent dissipation, respectively. The constant C_μ is set, as usual, to be 0.09[2]. Within the context of $k-\epsilon$ turbulence models, the transports of k and ϵ in the flowfield with a velocity vector $\vec{V}$ are governed by the following two respective differential equations, which are coupled to each other:

$$\frac{\partial \rho k}{\partial t} + \nabla \cdot (\rho \vec{V} k) = \nabla[(\mu_L + \frac{\mu_T}{\sigma_k}) \nabla k] + P_k - \rho \epsilon \quad (4)$$

$$\frac{\partial \rho \epsilon}{\partial t} + \nabla \cdot (\rho \vec{V} \epsilon) = \nabla[(\mu_L + \frac{\mu_T}{\sigma_\epsilon}) \nabla \epsilon] + \frac{\epsilon}{k}(C_{\epsilon_1} P_k + C_{\epsilon_2} P_\epsilon \quad (5)$$

The constants in the above two coupled equations are specified with the values given by $C_{\epsilon_1} = 1.44$, $C_{\epsilon_2} = 1.44$, $\sigma_k = 1$ and $\sigma_\epsilon = 1.3$ [2]. With the predicted values of k, the pressure variable p^* shown can be expressed by $p^* = p + \frac{2}{3}\rho k$. The turbulent production term shown in the source term of equation (4) is due to shear stress and is defined as $P_k = \mu_T \nabla \vec{V} (\nabla \vec{V} + (\nabla \vec{V})^T) - \frac{2}{3} \nabla \cdot \vec{V} (3\mu_T \nabla \cdot \vec{V} + \rho k)$.

While $k-\epsilon$ turbulent model has the property of offering robustness, economy and reasonable accuracy, it is performed poorly when applying it to the problems involving non-equilibrium boundary layers. It tends to under-predict the onset of separation and influence the prediction performance, at least, for the flow over an obstacle of various shapes. To resolve this problem, the shear stress transport (SST) turbulent model [3], implemented in Fluent, exploits the $k-\omega$ model at the wall and $k-\epsilon$ in the bulk flow. A blending function built in this commercial package ensures a smooth transition between the two models.

RESIDUAL DECAY ENERGY FOR LONG-TERM COOLING

In order to investigate the residual decay energy release rate for spent-fuel for long-term cooling of the reactor facility, we follow the work which have proposed in [4] to calculate the fission product decay power and heavy element decay heat. The fission product decay power after shutdown may be calculated as follows:

$$\frac{P}{P_o}(\infty, t_s) = \frac{1}{200} \sum_{n=1}^{n=11} A_n exp(-a_n t_s) \quad (6)$$

$$\frac{P}{P_o}(t_o, t_s) = (1+K)\frac{P}{P_o}(t_o, t_s) - \frac{P}{P_o}(\infty, t_o + t_s) \quad (7)$$

where $\frac{P}{P_o}$ is fraction of operating power, t_o is cumulative reactor operating time, t_s is time after shutdown, K is uncertainty factor, A_n and a_n are fit coefficients.
For heavy element decay heat, the following equations are used

$$\frac{P(U-239)}{P_o} = 2.28 \times 10^{-3} C \frac{\sigma_{25}}{\sigma_{f25}} [1 - exp(4.91 \times 10^{-4} t_o)][exp(4.91 \times 10^{-4} t_s] \quad (8)$$

$$\frac{P(N-239)}{P_o} = 2.17 \times 10^{-3} C \frac{\sigma_{25}}{\sigma_{f25}} \{1.007[1 - exp(-3.41 \times 10^{-6} t_o)] exp(-3.41 \times 10^{-6} t_s) - 0.007[1 - exp(-4.91 \times 10^{-4} t_o)] exp(-4.91 \times 10^{-4} t_s)\} \quad (9)$$

where C is conversion ration, σ_{25} is effective neutron absorption cross section of $U-235$ and σ_{f25} is effective neutron fission cross section of $U-235$. The reader can refer to [4] for more details. Notice that we will use the above equations to calculate the energy sources of the fuel-rods.

NUMERICAL METHOD FOR CALCULATING THE WATER HEIGHT OF SPENT FUEL POOL

There are three processes for the spent fuel pool under severe accident of LOCA or failure of RHR system. Firstly, The coolant will be heated until its boiling point due to the residual decay heat of the fuel-rods. It is followed by decreasing the water height of the spent-fuel pool. Finally, the water height will continuously decrease and the fuel-rod will be uncovered in the environment. At each process, we use different thermal dynamical equation to predict the water level. We summarize all the equations as follows

$$\text{Process 1} : Q = V \int_{T_1}^{T_2} \rho_w(T) C_p(T) dT \quad (10)$$

$$\text{Process 2} : \dot{Q}(t) = \rho_w(T) A_2 \frac{dh}{dt} h_{fg} \quad (11)$$

$$\text{Process 3} : \dot{Q}_2(t, h_e) = \rho_w(T) A_3 \frac{dh}{dt} h_{fg} \quad (12)$$

where Q is total residual heat, ρ_w is the density of water, C_p is specific heat, V is the total water volume of the spent-fuel-pool, $\dot{Q}$ is the residual heat rate when the fuel-rods are not uncovered, $\dot{Q}_2$ is the residual heat rate when the fuel-rods are uncovered at the height h_e, A_2 and A_3 are the cross sections of water surface of spent fuel pool at process 2 and process 3 and h_{fg} is the latent heat. In present study, we solve the Eqs. (6-12) by numerical method including shooting method and fourth-order Runge-Kutta numerical integrator to get the predicted water height.

NUMERICAL METHOD FOR TEMPERATURE DISTRIBUTIONS OF FUEL BUNDLES

In this study, we are aimed to explore the temperature distributions of the fuel bundles under two situations: normal operation and LOCA accident, as schematic in Fig. 2 and 3.The decay heat induced by the fuel-rod can also affect the coolant flow and may result in a quite different flow and temperature phenomenon. The coolant movement is mainly driven by the buoyancy force established in difference of the density.

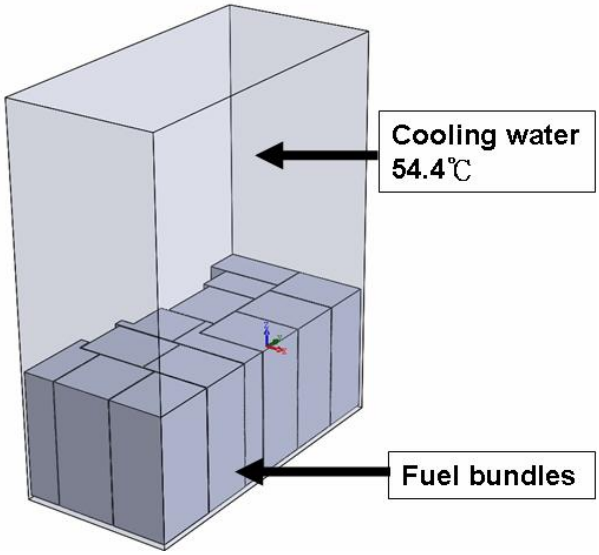

Figure 2 Schematic of the spent-fuel pool under the normal operation.

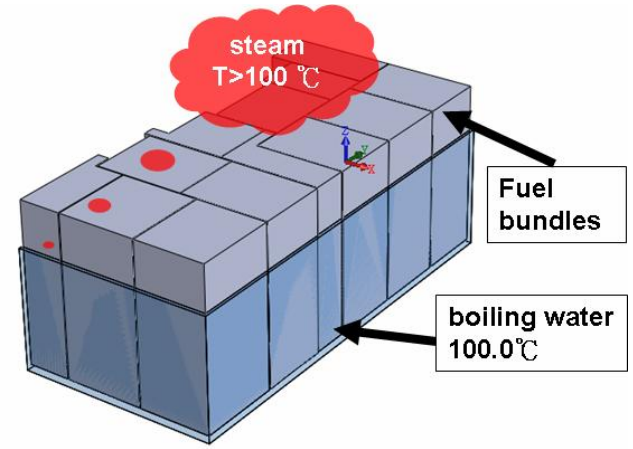

Figure 3 Schematic of the spent-fuel pool under the LOCA accident.

To predict the physically and geometrically complex flowfield, For the pre-processor, we use the SOLIDWROKS, to create the CAD (computer aided design) file for the investigated Atrium-10 fuel-bundle. By virtue of the embedded transformation Gambit IGES file in ANSYS Fluent, we can migrate the SOLIDWORKS CAD file to generate the surface meshes, followed by the interior mesh generation. With the generated meshes of tetrahedral and hexahedral types, which are used together so as to accurately resolve the boundary layer and predict flow separation/reattachment from the complex building surfaces, we can export these physically relevant mesh points to the flow solver, known as the ANSYS Fluent with the version of V6.3, to solve the primitive flow variables $\vec{V}$ and p, two turbulent quantities k and ϵ and temperature T.

ANSYS Fluent was developed within the control volumes for each working field variable in collocated grids distributed in the physical domain under investigation. The equations were discretized in a way to preserve the momentum, energy and turbulent quantities k, ϵ in each of control volume. To fulfill the continuity of mass, the well-known segregated SIMPLE (Semi Implicit Methodology for Pressure Linked Equations) solution algorithm of Patankar is used for calculating the pressure unknown. With the updated pressure solution computed from the PDE equation, one can replace its value to improve the initially assumed or previously calculated pressure in the momentum equation to calculate the new velocity components. The iterative procedures described above will be terminated until the computed maximum difference for solutions $\vec{V}$ and p between the two sets of consecutive solutions, cast in L_2 -norms, falls below the specified tolerance (10^{-7}). In ANSYS Fluent, the input modulus enable us to preserve the initial conditions, mesh arrangement for storing the field variables and convergence criteria. The boundary module is also provided for us to specify the boundary conditions for $\vec{V}$, T, k and ϵ. Nonlinear terms in the momentum equations will be linearized so as to render the algebraic equations. In present study, the algebraic multigrid scheme modulus will be chosen to solve all the algebraic equations.

RESULTS AND DISCUSSIONS

In this study, we assume the total volume of water in the spent fuel pool is 1740 m^3, the initial total decay heat power is 10.236 MWt. The cross sections for the process 2 and 3 are 144.7 m^2 and 86.8 m^2. The predicted temperature increasing rate is about 5.12 ^oC/hr. It leads the boiling time is about 11.52hr after failure of RHR system. After the water is boiling, the water height will decrease and vaporize. The fuel-bundles will then be uncovered to the environment. It is about 3days when the boiling happens. When the fuel bundles are uncovered in the environment, the cooling mechanism of the fuel bundles is changed from water cooling to vapor cooling. The decreasing rate of the water height will then be slow down. This can be explained that the decay heat of the fuel rods under the water level is decreasing when the fuel-rods are uncovered. We plot all the processes in Fig. 4 for the completeness.

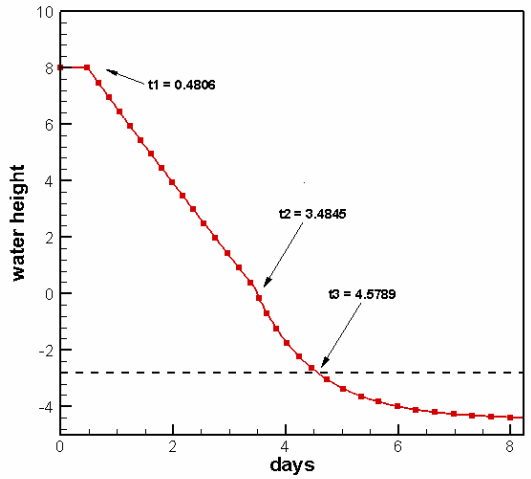

Figure 4 The predicted water-heights for under the LOCA accident. Note that t1 means the time when the water is at 100℃, t2 means the time when the fuel bundles start to uncover, t3 means the time then the fuel's temperature is at 1200℃.

After the analysis of the decay of the water height, we are now going to simulate the temperature distributions of the single fuel bundle under normal operation and LOCA accident. In this study, all the fuel rods in the fuel bundle are assumed as 169 inches.

For the normal operation case, the boundary conditions is as follows: for the interfaces of the solid-liquid are no-slip. the outer surfaces of the boron are periodic. In order to analyze the natural convection conserved, the inlet and outlet pressures are set to zero while the temperature is set to operation temperature (54.4^{o}C). The simulated results of temperature and velocity at outlet are shown in Figs. 5 and 6, respectively. From Fig. 5 and 6, it can be shown that the maximum temperature under normal operation is about 71.8 ^{o}C while velocity is about 0.218 m/s. the minimum temperature and velocity are located in the inner and outer water channel. This is due to the natural convection mechanism. When the difference of density is large, it will drive the fluid and convect the decay heat. At the corner, we can see the local lower temperature of the fuel rods due to the fact that there are less fuel-rod around itself and near the outer coolant water.

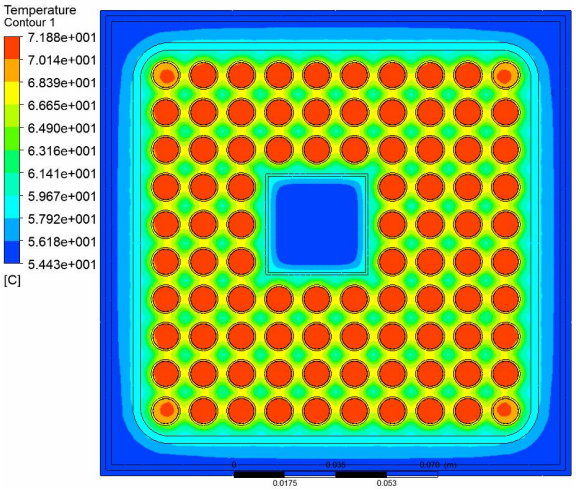

Figure 5 The predicted temperature contours for the single fuel bundle under the normal operation at outlet.

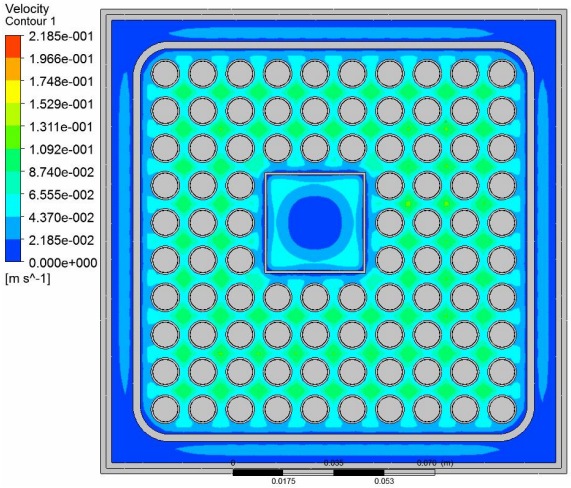

Figure 6 The predicted velocity contours for the single fuel bundle under the normal operation at outlet.

For the LOCA accident case, the boundary conditions is the same as the normal operation case expect that the temperature is set to boiling temperature (100^{o}C). The predicted results are shown in Figs. 7 and 8. In this case, the maximum temperature can be as large as 1532 ^{o}C. This is far away from the allowable temperature of the fuel rod, 1200 ^{o}C. Thus we should only consider the results which the maximum temperature of the fuel rod is under 1200 ^{o}C to avoid the water-metal reaction at exposure process, which is about 2.788 m from the Fig.9. By comparing Fig. 9 and Fig. 4, we can conclude that the allowable exposure time is about 4.5789 days.

We then plot the temperature and velocity in Fig. 10 and 11. Note that the velocity is about 3 m/s in this case, which is much larger than the normal operation case due to the fact that the coolant fluid is vapor. From Fig. 10, the natural convection of vapor flow make the energy be convected much faster and the temperature distributions are more uniform than the normal operation case in the fuel bundle. It is also noticeable that the velocity in the bundle are much faster than outside of the bundle. This is quite different with the normal operation case. This is due to the fact that the flow resistance of the bundle is larger than that of the outside bundle. The bundle thus can not be cooled efficiently via the vapor convection.

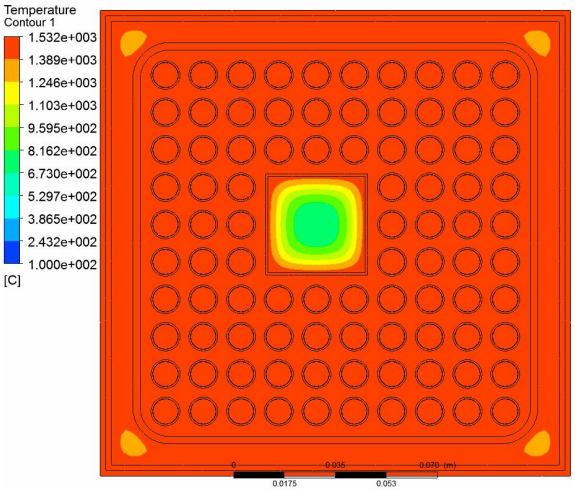

Figure 7 The predicted temperature contours for the single fuel bundle under the LOCA accident at outlet.

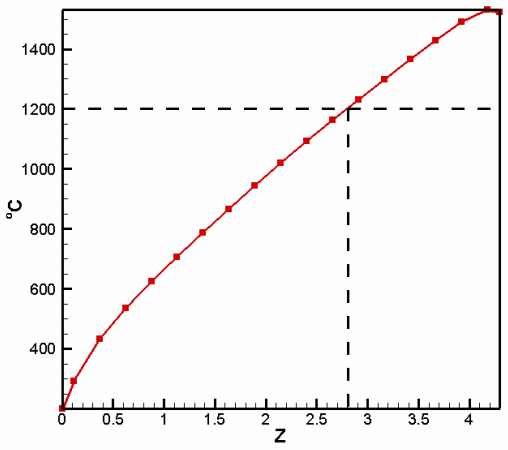

Figure 9 The predicted temperature for the single fuel bundle under the LOCA accident. The maximum allowable temperature (1200°C) is located at 2.788m uncovered length.

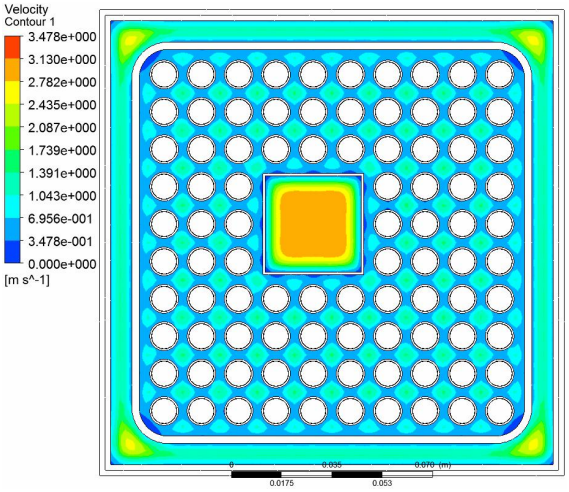

Figure 8 The predicted velocity contours for the single fuel bundle under the LOCA accident at outlet.

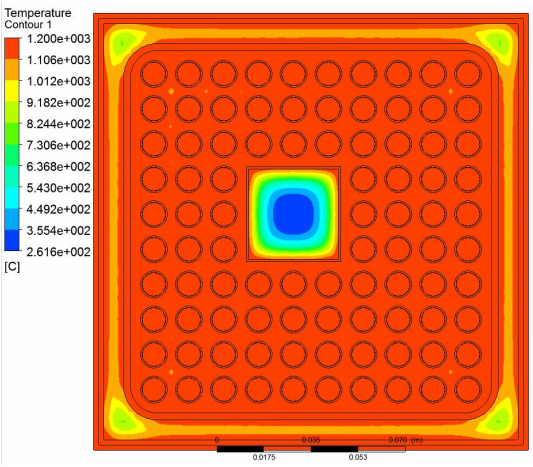

Figure 10 The predicted temperature contours for the single fuel bundle under the LOCA accident at 2.788m uncovered length.

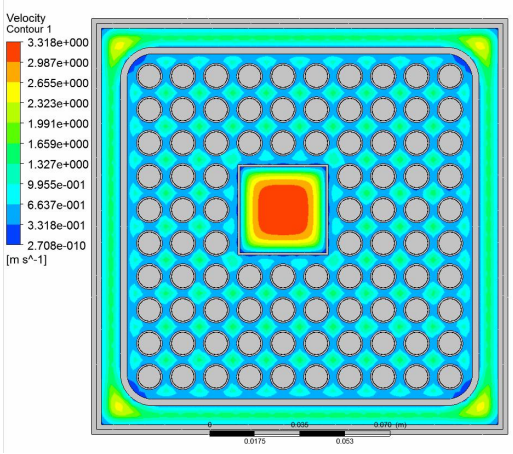

Figure 11 The predicted velocity contours for the single fuel bundle under the LOCA accident at 2.788m uncovered length.

CONCLUSION

The thermal hydraulic behaviour during the normal operation and the LOCA accident at spent fuel pool are numerically simulated in this study. The water height of the spent fuel pool was predicted by the thermal dynamical equation and residual decay energy equation. The ANSYS Fluent is used to simulated the temperature and velocity distributions for a single Atrium-10 fuel bundle. From the simulated results, we can predict the decreasing of water height of the spent fuel pool and the allowable exposure time. We also analyse the flow and thermal physics of the fuel bundle. The results confirm that the present framework can be used as a reliable analysis tools.

REFERENCES

[1] EMF-2577(P) Revision 0, Chinshan ATRIUMTM -10 Mechanical Design Evaluation Report and Chinshan Unit 2 Reload CS2-R18 ATRIUMTM -10 Mechanical and Thermal-Hydraulic Report, Framatome ANP Richland Inc., May 2001.

[2] Launder, B. E., Spalding D. B., The numerical computation of turbulent flows, Computational Methods in Applied Mechanics and Engineering, Vol. 3, 1974, 269-289.

[3] Menter, F. R., Zonal two equation $k-\epsilon$ turbulence models for aerodynamic flows, AIAA Paper, 93-2906, 1993.

[4] Branch Technical Position. (BTP) Auxiliary Systems Branch (ASB) 9-2