Proceedings of the World Congress on Engineering 2013 Vol III,
WCE 2013, July 3 - 5, 2013, London, U.K.

Numerical Simulation of Flow and Heat Transfer in Spent Fuel Cooling Ponds

Paul Robinson, Reaz Hasan* and Jenna Tudor

***Abstract*—Numerical simulation on a full scale model of a cooling pond for spent nuclear fuel has been conducted. The detailed flow and temperature field of the three dimensional domain reveals the interaction between buoyancy and convection dominated fluid streams. The effect of flow recirculation on the temperature distribution has been analysed. Parametric studies by varying the inlet flow and heat generation highlight that CFD can provide very useful insight into the flow development of such complex situations.**

***Index Terms*—Spent fuel, cooling pond, buoyancy flow**

I. Introduction

Spent nuclear fuel which is discharged from a reactor core remains highly radioactive for many years before it is ready for further reprocessing to allow permanent storage. During this long period fuel is kept immersed under water in huge enclosed ponds to stop radiation leakage and to provide a medium for heat transfer. The basic design and operation of these ponds are very similar irrespective of the country as is evidenced from the limited volume of literature [1]-[4] available on this topic. Recently, there appears to be a renewed interest amongst scientists, engineers and policy makers to scrutinise the safety issues of components of the nuclear industry including spent fuel pools after the Fukushima crisis and due to terrorist threats [1],[5]

These ponds can be thought of as huge water reservoirs which are more than hundred metres in length with a typical water level of about 8m. The bottom and side walls are constructed with heavy concrete, whereas the top surface of the pool is open allowing evaporation to take place. Some ponds are open totally to the atmosphere, others are covered by a ceiling on their building, both arrangements allow excess heat to be released to the environment. Depending on the type of spent fuel it can be stored as fuel rods which are enclosed in circular Multi-Element-Bottle (MEB) [2] or rectangular fuel containers. These are stacked under water on specially designed racks in such a way that they are always under more than 2m of water to stop radiation leakage. Water pumps maintain a continuous flow of water to avoid any risk of overheating and fuel stacking is designed to reduce eventual boiling in case of pump failure. Wang et al. [1] conducted a transient analysis under an accident case scenario to show that if the pump stops working conventional cooling ponds will take only 16.5 hours to reach a boiling point raising alarms for a crisis; they propose an alternative passive cooling system. Their calculations are based on bulk heating of the full pond water whereas practical experiences confirm that the temperatures may vary significantly from one position to another. In normal operating conditions, temperature variation within the pond is undesirable because it can cause some leakage of water through construction joints in the concrete walls/floor due to non-uniform thermal expansion.

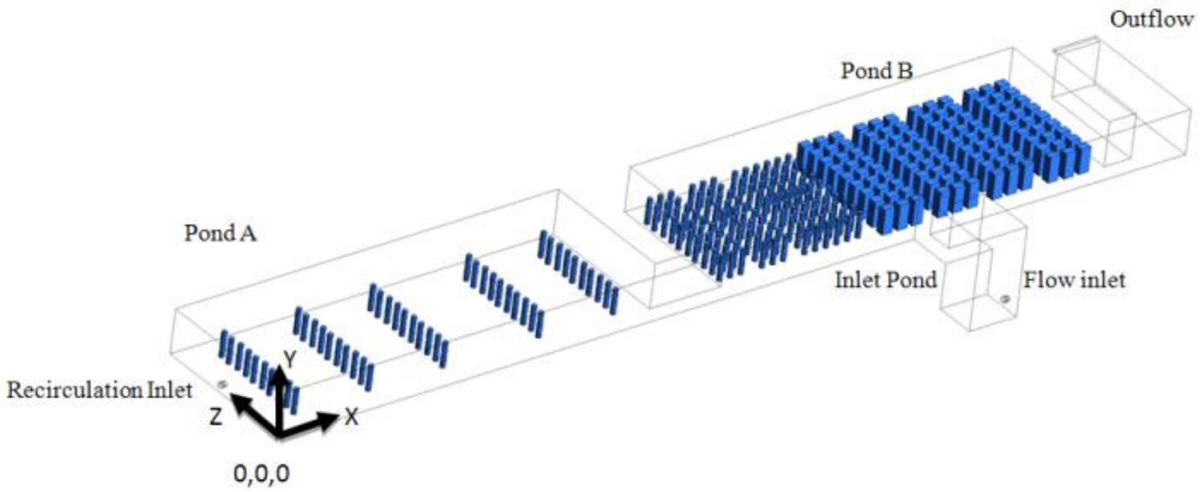

Fig. 1. The computational domain showing the three ponds, MEB's, Fuel containers and recirculation

Manuscript received March 10, 2013; revised April 4, 2013.

R. Hasan is with the University of Northumbria, Newcastle, NE1 8ST, UK (corresponding author to provide phone: 0191 243 7233; fax: 0191 227 3684; (e-mail: reaz.hasan@northumbria.ac.uk).

J. Tudor is with the University of Northumbria, Newcastle, NE1 8ST, UK (e-mail: jenna.tudor@northumbria.ac.uk).

P.Robinson was a student with the University of Northumbria, Newcastle, NE1 8ST.

Adding further complexity to this is that the fuel cladding needs to be protected from corrosion by caustic dosing and there is a lack of understanding of how the caustic dosing can be administered for optimum performance. So far the stacking pattern and most other operations are decided based on fuel data, reprocessing schedules and experience. Unfortunately very little information is available on the fundamental transport mechanism of heat and mass transfer in such ponds.

The objective of this work is employ the methodology of Computational Fluid Dynamics to establish a numerical model of a full scale cooling pond with a theoretical but realistic number of MEB's (2.5m height x 1m x 0.8m, weighing 2.5 t) and fuel containers (2.5m high and a radius of 0.5 m weighing 1 t) as shown in Fig. 1. The model is constructed of two ponds housing the spent fuels and an inlet pond for water supply. The figure also shows a possible re-circulation entry at plane X=0 and both cases were considered in this study.

Due to the large sizes involved (approx. 170m x 28m x 8m), the complexity of the flow and unavailability of reliable experimental data, a number of approximations were made in terms of boundary conditions. However, the presented methodology highlights that CFD can provide useful guideline for optimum stacking of fuel as well as to allow maintenance personnel to take well informed decisions about caustic dosing and data monitoring. The most important aspect of this paper is that it has attempted to address the fundamental transport mechanism that takes place in such full scale model which has not been reported.

II. Methodology

Calculations were carried out using the commercial CFD package of ANSYS FLUENT [6]. The methodology, which is fairly established and can be found in many text books such as Versteeg et al. [7], involves the iterative solution of the Navier-Stokes equations along with continuity and energy equation on collocated variables within non-uniform unstructured mesh configuration. In the absence of any other information, it was decided to incorporate the standard k-□ model given that the flow at the inlet pipe may be turbulent and also the Rayleigh number dominated buoyancy flow in-between the MEB's may also be turbulent for moderate temperature differentials [8]. Buoyancy forces were also activated during the solution. After trials on coarser meshes, the final grid was chosen to have 2 million cells and Fig. 2 shows the grid distribution on an X-Z plane.

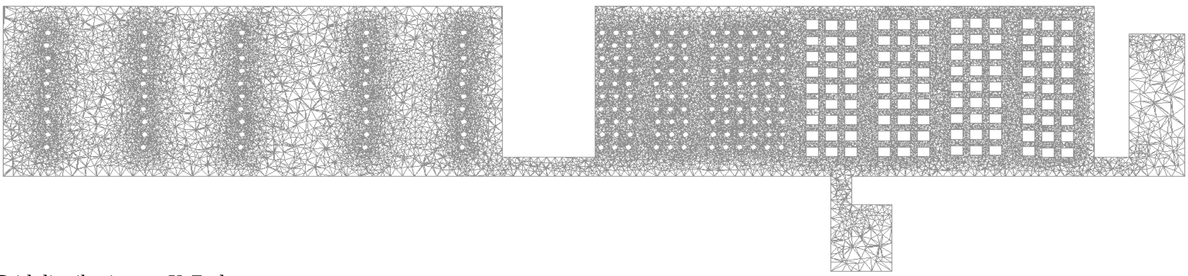

Fig. 2. Grid distribution on X-Z plane.

Although a rigorous grid dependency test was not carried out due to resource constraints, the data reported in this paper were found to be almost insensitive for two consecutive grid numbers and hence are believed to be free from any significant numerical uncertainty. The element metrics for mesh quality yielded 0.8 or more for 98% of the mesh. A typical run on a single Intel core 2Duo E6600 2.4 GHz processor took about 30 hours of computing time.

The inlet boundary condition was specified via a uniform velocity at the inlet pipe based on 243 kg/min flow rate and the outlet was specified as zero gradient which is well justified due to long distance from the zone of interest. Convection boundary conditions were used for the MEB surfaces where surface heat transfer coefficient was specified to be 16 W/m^2-K and heat generation of 1.04 kW/m^3 was specified based on actual data. The bottom and side walls were considered as no-slip walls with constant temperature of 10 ^{o}C and 12 ^{o}C respectively. The top surface which is 2.5 m above the fuels was considered as wall with a uniform temperature of 17 ^{o}C. These temperatures were selected by referring to the actual operational scenario but they are subject to further scrutiny and sensitivity analyses must be carried out for improved confidence. The most important reason for choosing the isothermal boundary condition was that it allows heat transfer through these surfaces and thus mimics the real situation qualitatively. To what extent these boundary conditions represent the actual situation is a matter for detailed investigation which is deemed to be beyond the scope of the present work. For the case of flow field with re-circulation, a second inlet condition was specified at the recirculation pipe as shown in Fig. 1.

III. Results and Discussion

A. Overall Flow Field

Fig. 3 shows the streamlines for the full three dimensional domain and depicts a very interesting flow field. Water that enters through the pipe at the bottom of the inlet pond can be seen to rise up in the form of 'twisted plume' which then takes a further turn spreading (to some extent) along the MEB's and finally adjusting itself with the outlet section of the pond. Fluid motions generated by buoyancy can be seen at few other locations along with adjacent shear layers which then interact with the primary flow stream. It is believed that the above motions, particularly away from the primary stream, are dictated by two factors, one due to the flow geometry which changes as soon as the stacking is changed and two the rate of heat release from the spent fuels.

B. Effect of Re-Circulation

When the outlet flow from pond B is re-circulated through the far end of pond A (on plane X=0), the flow field changes extensively. Fig. 4 shows a plot of fluid entering through pond A at mid-height of the recirculation pipe (Y=5 plane) where a continuous convection dominated fluid stream can be seen to extend from pond A to pond B.

ISBN: 978-988-19252-9-9
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

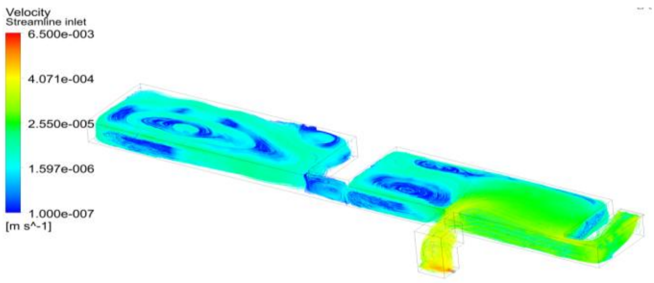

Fig. 3 Three dimensional streamlines for flow field without recirculation.

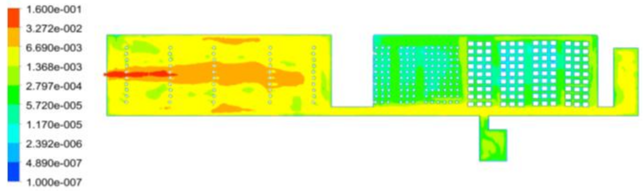

Fig. 4. Velocity magnitude contour with recirculation.

The differences in flow fields are very pronounced as shown in Fig. 4 by the temperature plots at different transverse positions of Z=1, 5, 10, 20 metres. The effect of recirculation in pond A appears to be much less pronounced than pond B which is possibly related to the smaller amount of heat release and also due to the absence of any dominant primary flow stream. However, the overall temperature level is higher when recirculation is switched off. This is due to an interesting interaction between fluid flow and heat balance and may be explained as follows. In Fig 5a, (recirculation switched off) higher temperature fluid diffusing from pond A mixes with primary stream but due to the fact that the amount of water flowing to the primary *stream* is very small and hence the total amount of heat transported with the flow, we can see the dominance of the MEB's on the temperature curves. The cooling effect of fresh water as well as flow dispersion in the Z-direction is also clearly visible from this graph. On the other hand, Fig 5b demonstrates that slightly cooler but higher magnitude of flow from pond A mixes with the primary stream and raises the temperature. A direct comparison of the temperature variations for flows with and without recirculation is shown in Fig. 5c.

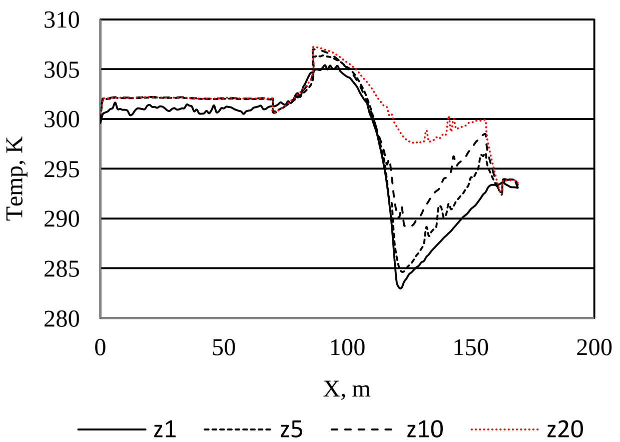

Fig. 5(a) Temperature variation in Z direction without re-circulation (Y=5m).

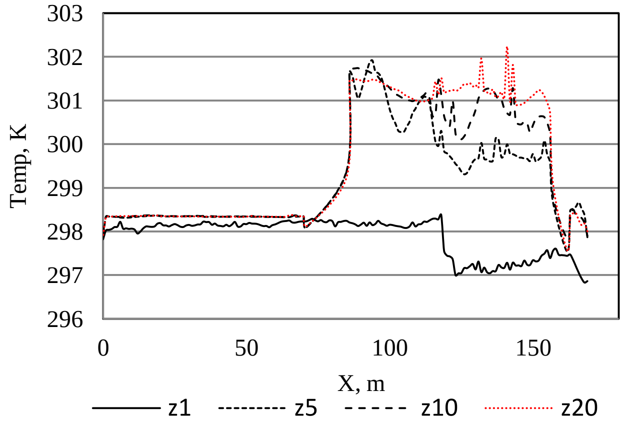

Fig. 5(b) Temperature variation in Z direction with recirculation (Y=5m).

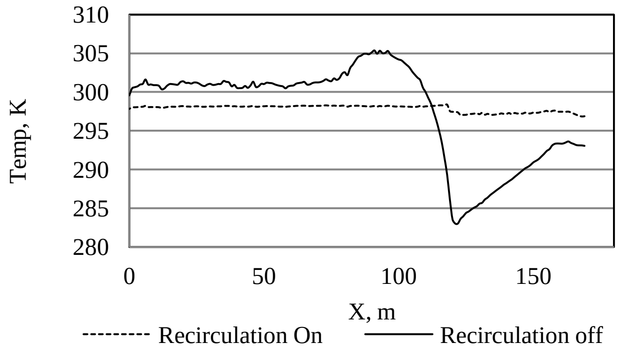

Fig. 5(c) Temperature variation with and without re-circulation (Y=5, Z=10).

C. Effect of Temperature Induced Buoyancys

The effect of buoyancy can be seen in most of the flow domain and as an illustration Fig. 6 shows the temperature contour across a vertical plane (Z=13m) in pond A without recirculation. The contours clearly show that the element surfaces have the highest temperatures and more importantly displays a vertical temperature gradient. To maintain continuity of flow, colder fluid from near the concrete base can be seen to be moving in an upward direction. The large gap between the water surface and the element is also clearly seen in this figure. This temperature gradient gives rise to a thermally stratified layer which serves as the primary potential for the sustained natural convection current which acts as the medium to expel heat in three different ways. These are (a) latent heat loss by evaporation from the top surface, (b) conduction loss through side wall and bottom concrete foundation and (c) convection by the primary stream. The isothermal boundary condition does allow heat transfer through these surfaces.

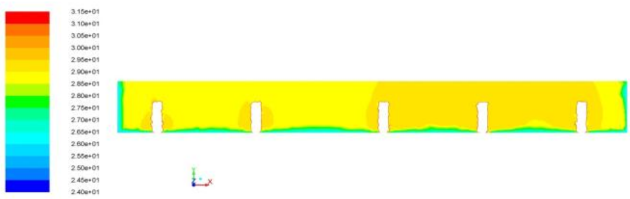

Fig. 6 Thermal stratification in pond 2 (without re-circulation).

D. Parametric Study

Once the CFD model is established, various parametric studies can be conducted on this flow domain which is a matter for further work. In this paper, we present the

ISBN: 978-988-19252-9-9
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

systematic analysis of the following two variations. We have studied the effect on fluid temperature by changing the amount of heat release by the fuel and by varying the flow rate at inlet when recirculation is switched off. The results are presented in Figs. 7a-b for three configurations where 100% means the standard test case considered previously. It can be seen that for the same flow rate, 50% reduction in heat release causes a temperature drop of about 7 ^{o}C in pond A, whereas 50% drop in flow rate causes a much less (~3 ^{o}C) temperature rise. Also the temperature distribution in pond A is more stable compared with pond B because the fluid motion in A is sustained by diffusion dominated thermal gradient only. On the other hand, the temperature variation in pond B demonstrates larger non-uniformity at Z=1 m plane (Fig. 7a) which gradually decreases towards the far end of the pond B in the transverse direction as can be seen in Fig. 7b. We present some sensitivity study data based on the flow and heat released and these are shown as follows. These test cases were selected based on the information from the industry that flow recirculation does allow better uniformity of temperature. No recirculation duct was created in the model to save the computational effort, but the recirculation flow rate was kept the same as the outflow. Sensitivity study was conducted by varying the flow rate and the heat generated. Case 1 represents flow conditions presented above without recirculation. For Case 2, flow unchanged but heat generation rate halved and for case 3 flow halved but heat generation kept the same.

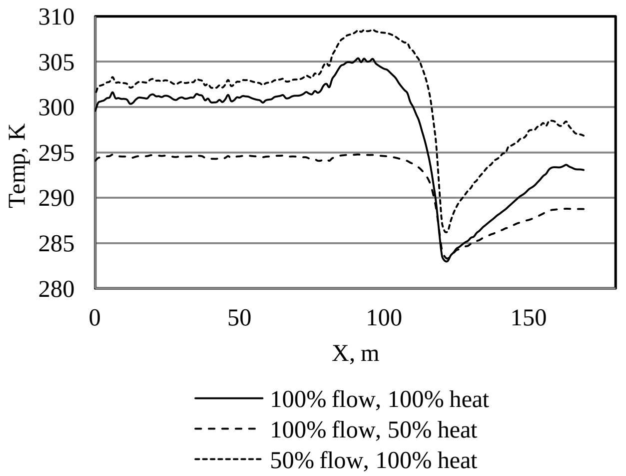

Fig. 7a Temperature variation with different flow and heat generation rates at Y=5, Z=1 plane.

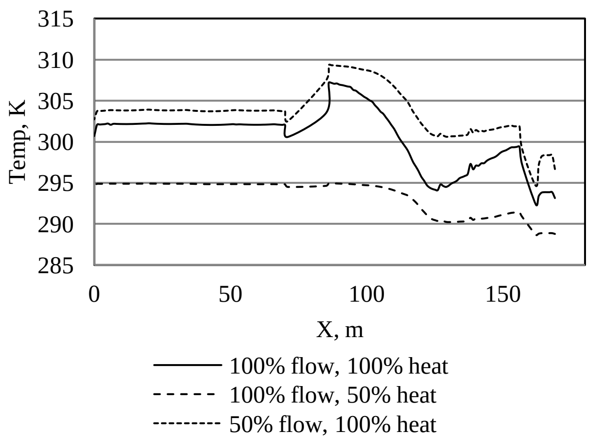

Fig. 7b Temperature variation with different flow and heat generation rates at Y=5, Z=15 plane.

IV. Conclusions and Further Work

Based on the work presented in this paper the following remarks can be made:

1) The thermal behaviour of the cooling pond can be explained by analysing the interaction between buoyancy driven diffusion and convection dominated fluid streams.
2) Recirculation has been found to have a big influence on the overall temperature distribution. Based on the results presented in this paper, it can be said that flow recirculation helps to attain a more uniform temperature.
3) More work needs to be done to improve the boundary conditions of the model. This can be achieved either by including part of the concrete structure within the solution domain and also by considering evaporation from top surface [9].
4) Experimental data with well documented operational and environmental conditions and seasonal variations are needed to make a full validation of the model.

References

[1] K. Wang, Y. Guo and He-yi Zeng, "Spent fuel pool transient analysis under accident case and the flow establishment process of passive cooling system," China located International Conference on Information Systems for Crisis Response and Management, Nov. 27-29, 2011.
[2] A. Jenkins, "Submerged water jet decontamination of multi-element bottles – 11521," WM2011 Conference, February 27- March 3, 2011, Phoenix, Arizona.
[3] E. G. Kudryavtsev and A. V. Khaperskaya, "Problems of spent nuclear fuel management in Russia and prospects for their solution," Russian J. of General Chemistry, vol. 81, no. 9, pp. 1928-1931, 2011.
[4] M. Molnar, L. Palcsu, Z. Major, E. Svingor, M. Veres and T. Pinter, "Dissolved gas measurements of the cooling ponds of Paks nuclear power plant, Hungary," J. Radiation Nucl. Chem., vol. 286, pp. 741-744, 2010.
[5] P. P. Predd, "Perils of plutonium," New Article, IEEE Spectrum, July, 2005.
[6] ANSYS FLUENT 13.0 (2010).
[7] H. K. Versteeg and W. Malalasekera, *An introduction to Computational Fluid Dynamics*. Pearson Education Limited, Harlow, England, 2007.
[8] D. A. Iyi, R. Hasan and R. Penlington,." Interaction effects between surface radiation and double-diffusive turbulent natural convection in an enclosed cavity filled with solid obstacles," Proc. of Advances in Computational Heat Transfer, ICHMT , Bath, UK, 2012.
[9] M. D. Hancock, "Indoor swimming pools and leisure centres – A model to improve operational effectiveness and reduce environmental impact," CIBSE Technical Symposium, Sept. 6-7, 2011, De Montfort University, Leicester, UK.

ISBN: 978-988-19252-9-9
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)