Annals of Nuclear Energy 114 (2018) 342–353

Contents lists available at ScienceDirect

Annals of Nuclear Energy

journal homepage: www.elsevier.com/locate/anucene

Zero-dimensional transient model of large-scale cooling ponds using well-mixed approach

Ahmed Ramadan *, Reaz Hasan, Roger Penlington

Northumbria University at Newcastle, Department of Mechanical and Construction Engineering, Newcastle upon Tyne, NE1 8ST, UK

ARTICLE INFO

Article history:
Received 18 July 2017
Received in revised form 9 December 2017
Accepted 14 December 2017

Keywords:
Spent fuel
Large-scale cooling ponds
Analytical modelling
Well-mixed approach
Transient heat transfer

ABSTRACT

Nowadays, nuclear power plants around the world produce vast amounts of spent fuel. After discharge, it requires adequate cooling to prevent radioactive materials being released into the environment. One of the systems available to provide such cooling is the spent fuel cooling pond. The recent incident at Fukushima, Japan shows that these cooling ponds are associated with safety concerns and scientific studies are required to analyse their thermal performance. However, the modelling of spent fuel cooling ponds can be very challenging. Due to their large size and the complex phenomena of heat and mass transfer involved in such systems. In the present study, we have developed a zero-dimensional (Z-D) model based on the well-mixed approach for a large-scale cooling pond. This model requires low computational time compared with other methods such as computational fluid dynamics (CFD) but gives reasonable results are key performance data. This Z-D model takes into account the heat transfer processes taking place within the water body and the volume of humid air above its surface as well as the ventilation system. The methodology of the Z-D model was validated against data collected from existing cooling ponds. A number of studies are conducted considering normal operating conditions as well as in a loss of cooling scenario. Moreover, a discussion of the implications of the assumption to neglect heat loss from the water surface in the context of large-scale ponds is also presented. Also, a sensitivity study is performed to examine the effect of weather conditions on pond performance.

1. Introduction

In the past decades, increasing the use of nuclear power for electricity generation has gained a lot of attention amongst scientists. Nuclear reactors around the world are now discharging a massive amount of spent nuclear fuel, which is predicted to reach approximately 445,000 t HM (metric tonnes of heavy metal) by 2020 (Zohuri and Fathi, 2015). This includes 69,000 t in Europe and 60,000 t in North America. Despite the recent incident at Fukushima, Japan (Kuo et al., 2011), nuclear power generation continue to grow in developed countries, as evidenced by the recent massive investment in nuclear energy by the UK government in approving an £18bn nuclear plant at Hinkley Point C. This will deliver 7% of Britain's electricity needs for the next six decades (UK Government, 2016).

The issue of long-term storage was not considered when the original decisions were made regarding the fuel cycle (Kozlov et al., 2000). Recently, waste management has become one of the major policy issues in most nuclear power programmes. Meanwhile, the options chosen for waste management can have extensive effects on political debates, propagation risks, environmental threats, and economic costs of the nuclear fuel cycle. This increases the significance of modelling the cooling ponds and analysing their performance to provide a better understanding of their pond thermal behaviour. This will allow for better operation and could offer mitigation options whenever needed in accident scenarios.

Several research investigations have considered the thermal-hydraulic behaviour of the spent fuel cooling ponds, which are mainly focused on accident scenarios and their consequences (Kuo et al., 2011; Ahn et al., 2016; Chen and Yuann, 2016; Fu et al., 2015; Wang et al., 2012). These studies used two main modelling approaches. The first approach is the use of so-called system codes such as RELAP, TRACE, ATHLET, MELCOR and ASTEC. These codes are based on dividing the system into a network of pipes, pumps, vessels, and heat exchangers. Mass, momentum and energy conservation equations are then solved in one-dimensional form. Many phenomena and physical behaviour such as two-phase flows and pressure drop due to friction rely on empirical correlations. These codes are suitable for systems that can be represented by one-dimensional flows. However, when such a system involves multi-dimensional phenomena, these codes

* Corresponding author.
E-mail address: ahmed.ramadan@northumbria.ac.uk (A. Ramadan).

https://doi.org/10.1016/j.anucene.2017.12.043
0306-4549/

Nomenclature

A	surface area (m^2)
C_p	specific heat capacity at constant pressure (J/kg K)
C_w	specific heat capacity of water (J/kg K)
h_c	convection heat transfer coefficient (W/m^2 K)
h_{con}	condensation mass transfer coefficient (m/s)
h_{ev}	evaporation mass transfer coefficient (m/s)
$h_v(T)$	enthalpy of vapour at a given temperature (kJ/kg)
h_{fg}	latent heat of vaporisation for water (kJ/kg)
k	thermal conductivity (W/m K)
m	mass (kg)
$\dot{m}$	mass flow rate (kg/s)
M	molecular weight (kg/kmol)
N	mole number (kmol)
$\dot{N}$	molar flow rate (kmol/s)
Nu	Nusselt number
P	pressure (Pa)
$\dot{Q}$	heat transfer rate (W)
Ra	Rayleigh number
RH	relative humidity (%)
R_o	universal gas constant (J/K kmol)
Sh	Sherwood number
T	temperature (K)
V	volume (m^3)
x	wall thickness (m)
y	mole fractions
Δt	time step size (s)

Greek symbols

ε	emissivity
ρ	density (kg/m^3)
σ	Stefan-Boltzmann constant (W/m^2 K^4)

Subscripts

a	dry air
∞	ambient
c	convection
con	condensation
d	heat load
D	designed value
ev	evaporation
h	hall
l	leakage
m	make-up
p	pond
r	radiation
R	rack
sat	saturation
t	total
v	vapour
$vent$	ventilation
w	water
wb	wet bulb

do not provide a good approximation. Some attempts have been made to improve their capability to handle multi-dimensional flows. One of these attempts considers the system as an array of parallel one-dimensional pipes, where the interaction between them is allowed through cross-flow coupling. Although they provide improved approximations compared with purely one-dimensional approaches, these models do not offer appropriate descriptions of multi-dimensional flows. The MARS code is an example of attempts to include a multi-dimensional analysis capability in system codes (Tanaka, 2012).

The second approach is a numerical method such as computational fluid dynamics (CFD) which in principle can address details of thermos-fluid phenomena in cooling ponds Numerical methods such as CFD can be used, in principle, to address fluid flow and heat transfer scenarios in three dimensions using computers. The CFD methodology is now well-established, but the available literature indicates that a full CFD model of a spent fuel cooling pond may be not practically possible. This is due to their large size and the existence of complex phenomena, such as evaporation, which requires multiphase flow models. However, some studies have reported CFD modelling of spent fuel ponds taking into account only the water body without considering the humid air zone above or ventilation and their effect on the evaporation rate. Also, some of the challenges encountered during the CFD simulation have been discussed in our previous work (Hasan et al., 2015). An example of the use of CFD in improving the safety of such cooling ponds can be found in a study conducted by Ye et al. (2013), in which a new passive cooling system was designed to provide an adequate cooling for the CAP1400 spent fuel pool in emergency situations. Hung et al. (2013) used the CFD approach to predict the cooling ability of the Kuosheng spent fuel pool and to confirm that the existing configuration can provide enough cooling to meet licensing regulations with a maximum water temperature of 60 °C. A unique aspect of their work is that they used CFD in a more advanced way than in other studies to predict local boiling within the pool water, reflecting the strength of the CFD approach. Another use of CFD is to study flow characteristics within fuel assemblies. For example, a study conducted by Chen et al. (2014) investigated flow and heat transfer within a rod bundle using a three-dimensional model.

Yanagi et al. (2012a) produced a CFD model for a cooling pond and compared the predicted water temperature with those for the cooling pond at Fukushima Daiichi Nuclear Power Station under loss of cooling conditions. The water surface was modelled using a previously derived heat transfer correlation by the same authors (Yanagi et al., 2012b). The CFD model produced by Yanagi et al. (2012a) was further used to form a baseline for an analytical model "One-Region model" also generated by Yanagi and Murase (2013), Yanagi et al. (2016). This One-Region treats the water as on node with a single temperature value without taking into considerations its distribution. After that, they have examined the effect of the distribution of the heat load on the variation of water temperature and it was confirmed that the One-Region model applicable to predict the water temperature in the cooling pond during the loss of cooling scenario.

On the other hand, most of the studies adopting the system codes were concerned about investigating accident scenarios and their consequences. Carlos et al. (2014) used the TRACE best estimate code to analyse the safety of the Maine Yankee spent fuel pool. Ognerubov et al. (2014) investigated scenarios of the loss of water in a spent fuel pool in the Ignalina NPP using various system codes to identify potentially unrealistic parameters while performing the calculations. Groudev et al. (2013) used RELAP5 to study the thermal-hydraulic behaviour of spent fuel for a dry out scenario while transferring fuel from the Kozloduy NPP reactor vessel to the cooling pool. Additional studies dealing with fuel ponds can