IOP Conf. Series: Materials Science and Engineering **298** (2018) 012030 doi:10.1088/1757-899X/298/1/012030

Comparative study between single core model and detail core model of CFD modelling on reactor core cooling behaviour

R Darmawan

Malaysian Nuclear Agency, Bangi, 4300 Kajang, Selangor, Malaysia

roslid@nm.gov.my

Abstract. Nuclear power industry is facing uncertainties since the occurrence of the unfortunate accident at Fukushima Daiichi Nuclear Power Plant. The issue of nuclear power plant safety becomes the major hindrance in the planning of nuclear power program for new build countries. Thus, the understanding of the behaviour of reactor system is very important to ensure the continuous development and improvement on reactor safety. Throughout the development of nuclear reactor technology, investigation and analysis on reactor safety have gone through several phases. In the early days, analytical and experimental methods were employed. For the last four decades 1D system level codes were widely used. The continuous development of nuclear reactor technology has brought about more complex system and processes of nuclear reactor operation. More detailed dimensional simulation codes are needed to assess these new reactors. Recently, 2D and 3D system level codes such as CFD are being explored. This paper discusses a comparative study on two different approaches of CFD modelling on reactor core cooling behaviour.

1. Introduction

The design and development of a nuclear reactor has always been focused on its reliability to operate safely throughout its lifetime. The early nuclear reactor designs are mostly dependent upon experimental analysis [1], where complex configuration and various phenomena have to be investigated. This approach would require very specialised experimental facilities which may not be affordable to many researchers or institutions [2][3]. Later, the development of computational method has brought about new approaches on the design and analysis of nuclear reactor safety. This is where the usage of numerical method was employed for most of the nuclear safety assessment activities. The one dimensional simulation codes designed specifically for nuclear reactor safety analysis were then developed by various nuclear engineering institutions specifically to simulate the neutronics and thermal-hydraulic behavior of the reactors [4]. These one dimensional simulation codes have been extensively used since the last 3 decades by the nuclear industry in the nuclear reactor design process [5]. However, the complexity of the recent advanced nuclear reactor design requires three dimensional capabilities to investigate specifically localised phenomenon in the nuclear reactor. Thus, the last decade has shown new interest on using three dimensional simulation codes to analyse nuclear reactor performance and safety designs [3].

For most of the reactor design process and safety analysis, 1D simulation codes are used extensively due to its maturity and abundance usage experience. However, for the last two decades, 3D Computed Fluid Dynamics codes such as FLUENT and CFX have started to be used in addition to

1D simulation codes. The 3D capability is needed in certain cases where 1D code might not be able to predict. Most of the applications are in the Pressurised Water Reactors (PWR) which have more complex design and operating condition compared to Light Water Reactors (LWR). Among the applications are the study on sub cooled boiling phenomena, void fraction distribution, slugging, counter current flow limitation and pressurized thermal shock [6][7]. Although the operation and design of LWR reactors are less complicated, 3D CFD code is still needed in certain cases. Most of the works in LWR reactors, which employing CFD method are related to the emergency analysis of the reactors [3,7,8,9].

CFD study on reactor operational behaviour may involve the modelling of the reactor core as the heat source. 3D CFD modelling would require 3D configuration of the reactor core. Modelling the details of core configuration may incur heavy computational load as well as time consuming. Most CFD studies employ the simplification model to reduce the computational load and simulation time, especially during preliminary analysis [10][11][12][13][14]. Among the typical methods used are the simplified core approach and porous media approach. The simplified core approach would either simplified the core geometric configuration [11][14] or assuming the core as a single element [12]; whereas for porous media approach requires the input on porous media parameters obtained from the empirical study of the actual core [10][13]. In this study, the aspect of model configuration in CFD was investigated, two different model configurations were modelled and the results were compared to study the effect on cooling behaviour of a hypothetical reactor core.

2. Methodology

A typical open pool reactor core was modeled using CFD software FLUENT. The reactor is of cylindrical type with the top exposes to the ambient. The wall was assumed to be adiabatic so as to simulate the typical concrete shielding of an open pool reactor. Two different core configurations were modelled as shown in figure 1. The core of the reactor was modeled as a single cylinder and as an array of rod fuel elements. The two different core models were subjected to the same heat flux, which is expected to undergo cooling process through conduction and convection of the cooling medium. The model is assumed to be in steady state condition to simulate a normal operation of the reactor. The cooling fluid is normal water, operating at pool bulk temperature and at atmospheric pressure. No external force or pressure is imposed on the fluid, thus, it is assumed to be in stationary and in laminar flow condition. The pool wall is considered smooth and no frictional force is exerted on the fluid. The top part of the model is considered as convective wall, exposed to free stream air at room temperature. The convective heat transfer coefficient in free stream air is set to $10 W/m^2 K$ as used by *Rodriguez et. al.* [15] and Gastelurutia *et. al.* [16]. Other fluid properties of the model are shown in figure 2. The validation of the models was conducted on modeling program verification and numerical solution verification aspects as described in [17].

3. Results and discussion

The results of the simulation on both models with adiabatic wall condition are shown in figure 3 and figure 4. Both core models were subject to $3.5 W/m^2$ heat flux. The detail core model shows that the maximum temperature on the surface of the fuel core is about 127^oC, which is a typical maximum surface temperature of low power open pool reactor [18]. However, the result on the single core reactor shows much lower surface temperature at about 63^oC. It may resulted from the smaller surface area in the single core model that at the same heat flux per unit area, would produce much lower heat from the core surface. Both core models should be undergoing almost the same rate of heat dissipation process that the model with detail core and higher total heat, ended up with higher core surface temperature.

IOP Conf. Series: Materials Science and Engineering **298** (2018) 012030 doi:10.1088/1757-899X/298/1/012030

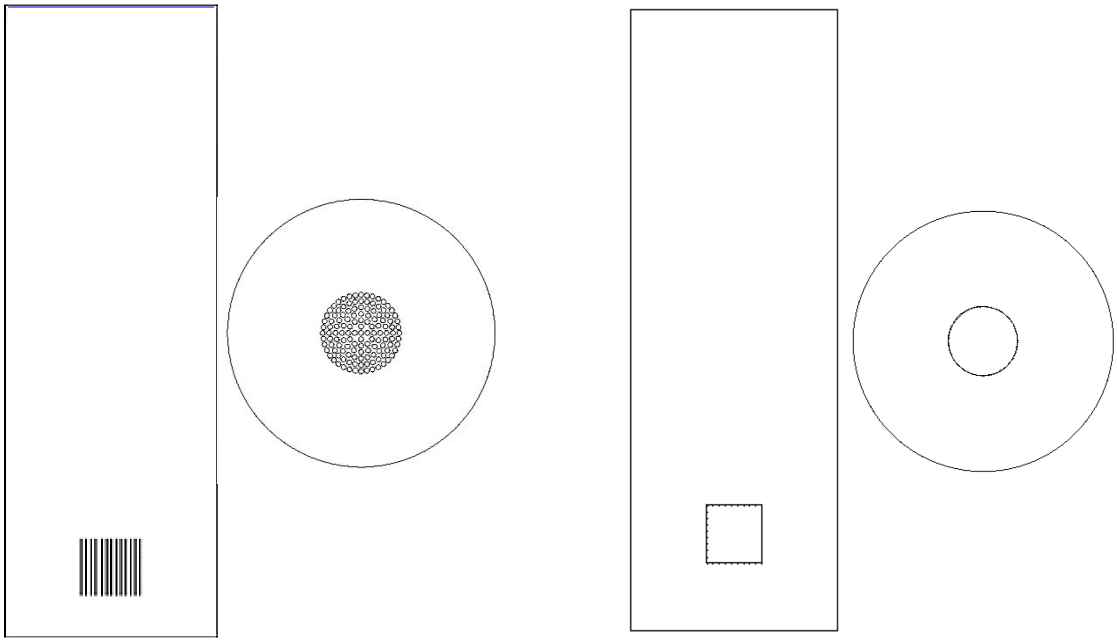


Figure 1. Detail core model (left) and single core model (right) of the reactor pool.

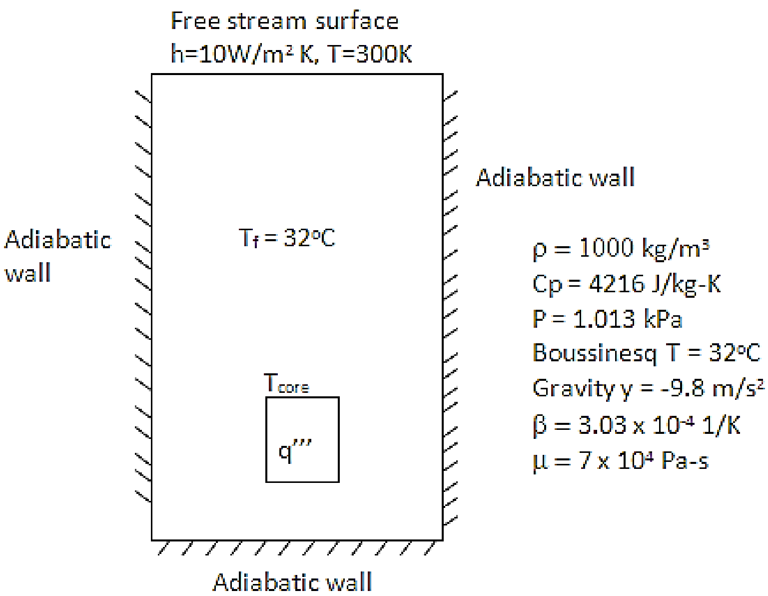
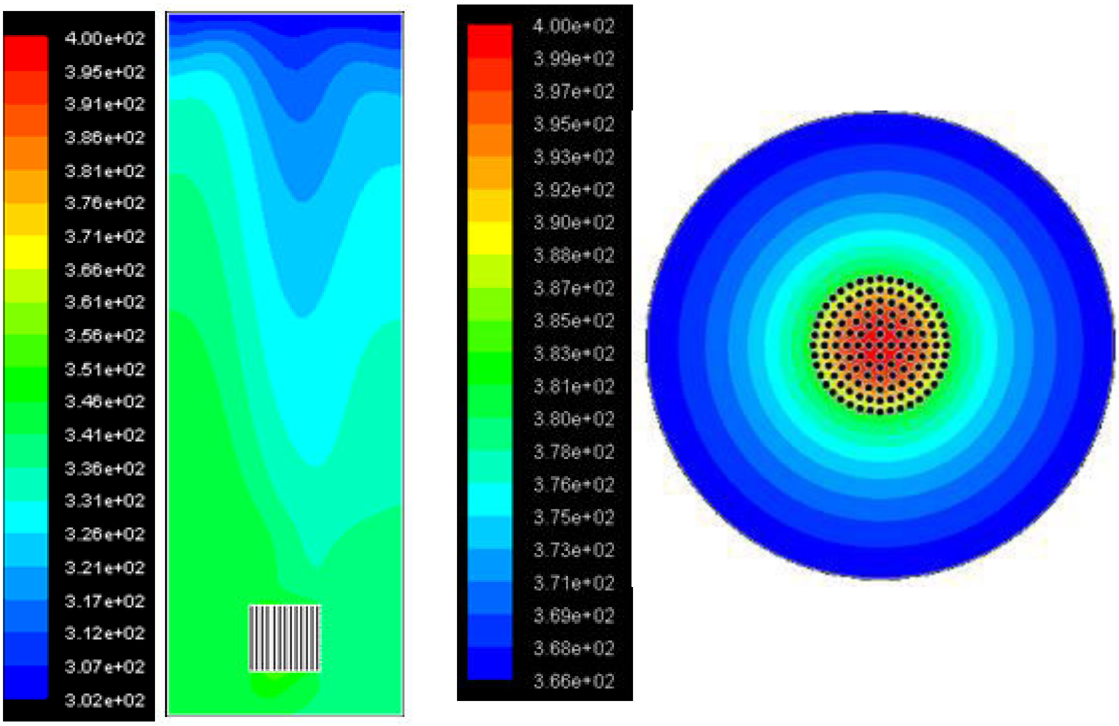


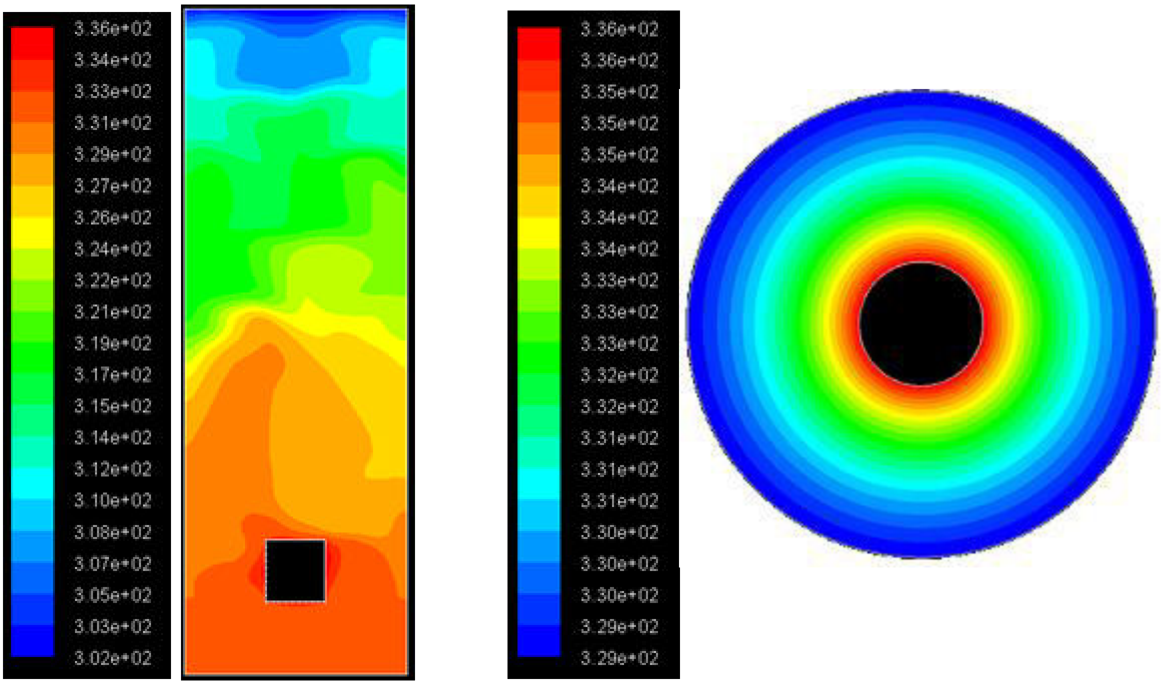
Figure 2. Schematic of the reactor pool model.

IOP Conf. Series: Materials Science and Engineering **298** (2018) 012030 doi:10.1088/1757-899X/298/1/012030



Vertical view Top view

Figure 3. Temperature profiles (in K) of detail core model with 3.5W/m^2 heat flux and adiabatic wall.



Vertical view Top view

Figure 4. Temperature profiles (in K) of single core model with 3.5W/m2 heat flux and adiabatic wall.

The stream function profiles illustrated in figure 5 shows distinct features, where detail core model has lower stream mass. Most of the recirculation stream occurs around the core with longer recirculation stream above the core and along the cylinder until the top of the cylinder. Two distinct convection modes could be seen that are the rigorous stream around the core and the more stable stream above the core. Whereas, for single core model, the same recirculation mechanism occurs around the core, however the convection streams distributed into three flow behaviours; rigorous around the core, mixed mode at the middle and more stable stream at the top. These features show that the detail core model has higher temperature gradient between the core and the ambient. The maximum mass stream is lower and the convection mechanism occurs in a stable manner up to the top of the pool. In the single core model, the significant temperature gradient may only be up to the middle of the pool where the mixed mode stream occurs. These convection mechanisms substantiated the discussion that the detail model may have higher surface heat due to more surface area, compared to the single model case. From the above results and discussion, the simple model such as single cylindrical core may not be a good representation of the actual core. A more detail modelling such as the detail core model would be a better representation of the actual reactor cooling behaviour as shown by Figure 3, where the results is comparable with the typical maximum core temperature of low power open pool reactor [18].

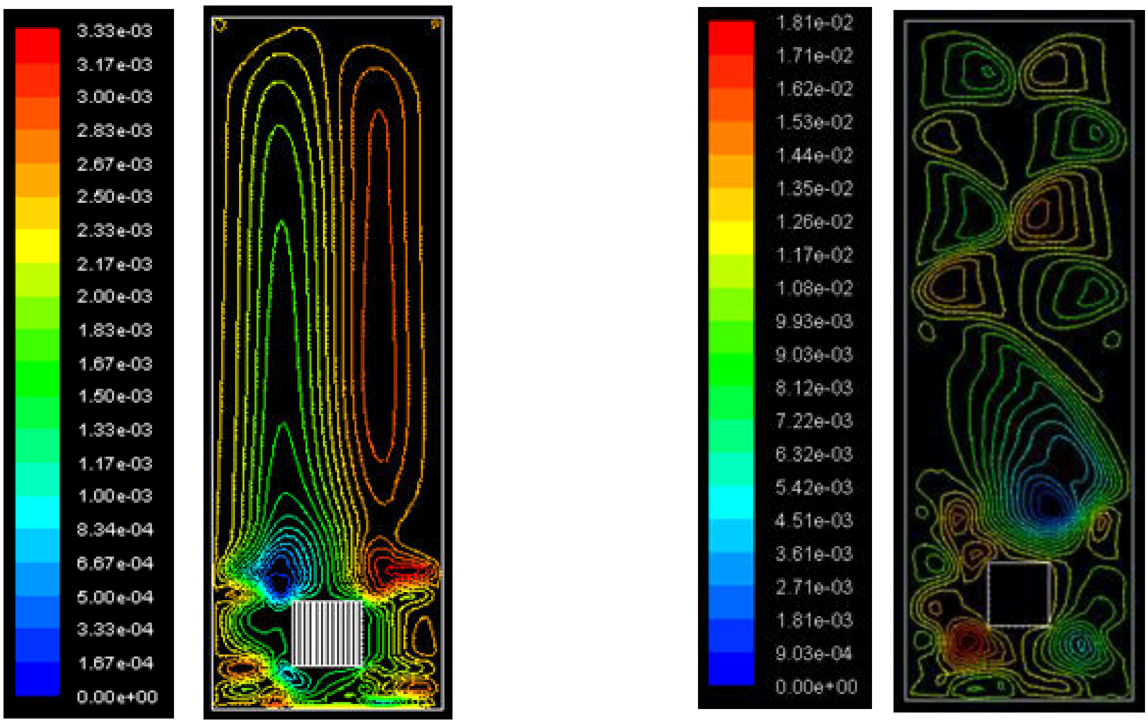


Figure 5. Stream function profiles (in kg/s) for full core model (left) and single core model (right).

Another comparative study was also conducted between the adiabatic wall and fixed wall cases. Both models were subjected to the same heat flux with the pool wall either set as adiabatic or fixed temperature at 300K. The results of the simulations are shown in figure 6 and 7. Both models show the same characteristics where the models with fixed temperature have lower core surface temperature compared to models with adiabatic walls. The models with adiabatic walls have only the top opening as the region of low temperature, where it is set as a free stream with temperature at 300K. This will reduce the heat dissipation capability when compared to the fixed wall temperature. In the fixed wall temperature case, low temperature region is on the surrounding wall as well as the bottom surface of the pool. Thus, heat would be dissipated quickly to the surrounding wall and the floor of the pool.

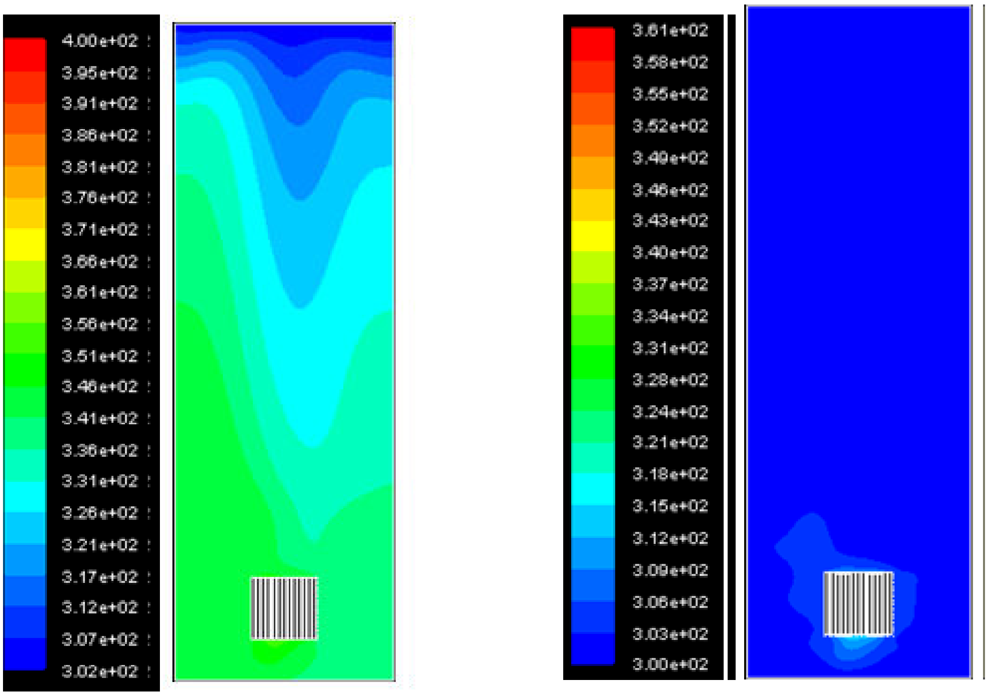


Figure 6. Temperature profiles (in K) of detail core model with adiabatic wall (left) and fixed wall temperature (right).

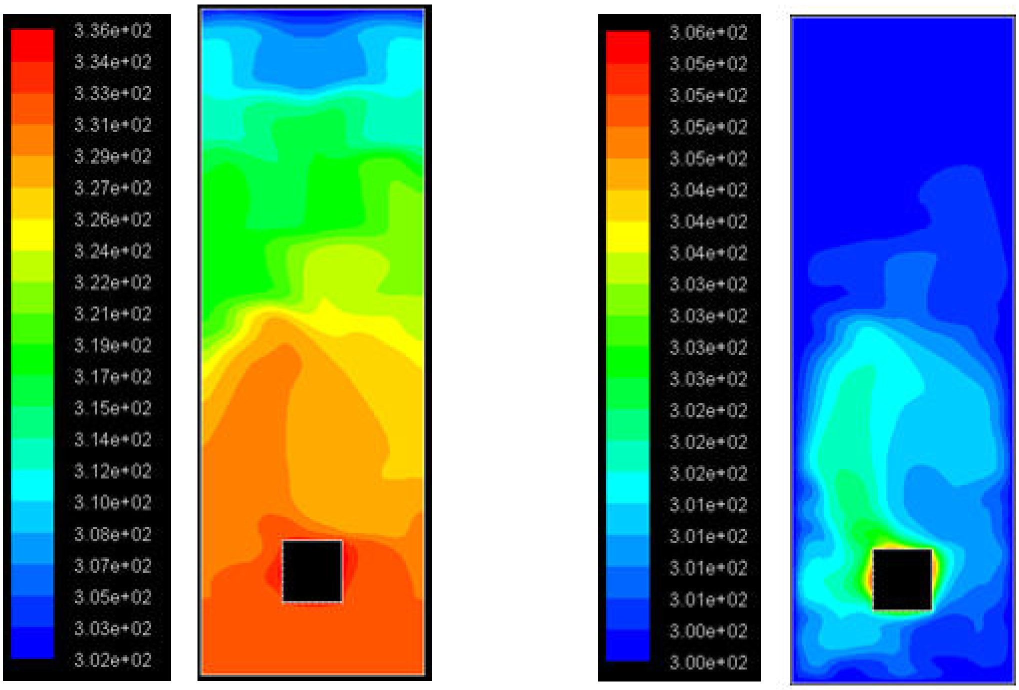


Figure 7. Temperature profiles (in K) of single core model with adiabatic wall (left) and fixed wall temperature (right).

The stream function profiles show similar characteristics for both cases where in the case of detail core model, the stream has two modes and the single core model has three stream modes as discussed earlier. This phenomenon signifies that both the adiabatic wall and fixed wall temperature undergoing the same heat convection mechanism in both models, and the amount of low temperature region will determine the heat dissipation capacity. The results of this comparative study also reveal that the adiabatic wall model is a better representation of the actual reactor pool wall compared to fixed wall temperature model. The adiabatic wall in detail core models resulted in 127^{o}C of maximum core temperature; whereas the fixed wall temperature model has only 88^{o}C maximum core temperature, which is much lower than the typical maximum core temperature for low power open pool reactor.

4. Conclusion

CFD modeling of a reactor core cooling behaviour would be more representative for a detail core model compared to a single core model. The detail core model produced similar characteristics of temperature and flow profiles to the typical low-power open-pool reactor as compared to the single core model. CFD modeling with adiabatic wall model is a better representation to simulate reactor core cooling with concrete shielding wall compared to fixed wall temperature model. The adiabatic wall model produced temperature and flow profiles which have close characteristics with the typical open pool reactor. A comparison with porous media approach may also be pursued to further evaluate the accuracy of each model.

5. References

[1] Bestion D 2004 Recommendation on use of CFD codes for nuclear reactor safety analysis *5th EURATOM Framework Program 1988-2002, Key Action: Nuclear Fission* (European Commission)

[2] Sardain P, Ayrault L, Laffont G, Challet F, Marie L B, Merrill B, Porfiri, M T and Caruso G 2005 The EVITA programme: experimental and numerical simulation of a fluid ingress in the cryostat of a water-cooled fusion reactor *Fusion Engin. and Design* **79** 1265–69

[3] Andreani M, Haller K, Heitsch M, Hemström B, Karppinen, I, Macek, J, Schmid, J, Paillere, H and Toth, I 2008 Benchmarking of CFD codes for application to nuclear reactor safety *Nuc. Engin. and Design* **238**, Issue 3 502-13

[4] Yadigaroglu G 2005 Computational fluid dynamics for nuclear applications: from CFD to multi-scale CMFD *Nuc. Engin. and Design* **235** 153-64

[5] Gao S, Leslie D C and Hewitt G F 2008 Improvements to the modelling of two-phase flow and heat transfer in a transient nuclear reactor analysis code *Appl. Therm. Engin.* **28** 915-22

[6] Koncar B, Krepper E and Egorov Y 2005 CFD modelling of subcooled flow boiling for nuclear engineering application *Inter. Conf. Nuclear Energy for New Europe 2005*, *Bled, Slovenia, September 5-8, 2005* p140.1-140.14

[7] Deendarlianto H T, Apanasevich P, Lucas D, Vallée C and Beyer M Application of a new drag coefficient model at CFD-simulations on free surface flows relevant for the nuclear reactor safety analysis *Annals of Nuclear Energy* **39(1)** 70-82

[8] Bernard J A and Lin W H 2001 Numerical model of MIT research reactor *IEEE Transc. on Nuc. Science,* Vol **8** p27-3 27-10

[9] Vanderhaegen M, Vierendeels J and Arien B 2001 CFD analysis of the MYRRHA primary cooling system *Nuc. Engin. and Design*, Vol **241** Issue 3 775-84

[10] Yan Y, Rizwan-uddin and Sobh N 2005 CFD modeling on reactor core *Math. and Computation, Supercomputing, Reactor Phys. and Nuc. and Bio. Applications*, *Palais des Papes, Avignon, France, September 12-15, 2005* (American Nuclear Society, LaGrange Park, IL).

[11] Khanbabaei B, Ghisemizad A and Farajollah H 2008 CFD calculation of fluid flow in VVER-1000 Reactors *Jou. of Appl. Sciences*, Vol **8(5)** p780-87

[12] Prosicky M and Zacha P 2003 Modeling of temperature field in a reactor vessel downcomer during transient *Transaction of the 17th Inter. Conf. on Structural Mechanics in Reactor Tech. Prague, Czech Republic, August 17-22, 2003*

[13] Sipaun S M 2014 Thermal-fluid modeling of the Missouri S&T Reactor *Doctoral Dissertations* **2358** (Missouri University of Science & Technology)

[14] Keshmiri A 2010 Thermal hydraulic analysis of gas cooled reactor flows *Ph.D. thesis* (University of Manchester)

[15] Rodríguez I, Castro J, Pérez-Segarra C D and Oliva A 2009 Unsteady numerical simulation of the cooling process of vertical storage tanks under laminar natural convection *Inter. Jou. of Therm. Sci.* **48** p708–21

[16] Gastelurrutia J, Ramos J C, Larraona G S, Rivas A, Izagirre J and Luis D R 2011 Numerical modeling of natural convection of oil inside distribution transformers *Appl. Therm. Engin.* **31** p493-505

[17] Rosli D, Nor Mariah A, Noraini A A, M Khairol Anuar M A 2017 Investigation on alternative method to enhance the cooling capacity of an open pool research reactor *AIP Conference Proceeding 1799* Vol **10:1** p28-38.

[18] Mesquita A Z 2005 Experimental investigation on temperatures distributions in a nuclear research reactor TRIGA IPR-R1 *Ph.D Thesis* (Universidade Estadual de Campinas, São Paulo, Portuguese)