Proceedings of the 18th UK Heat Transfer Conference
9-11 September 2024, Birmingham
UKHTC2024

SCOPING ANALYSES FOR THE DEFINITION OF NEW EXPERIMENTS FOR NATURAL CONVECTION AT HIGH RAYLEIGH NUMBERS.

J. Uribe[1*], S. Benhamadouche[1], A. Morente[1], P. Rotach[1]., J.-L. Vacher[2], M. S. Chae[3], D. Paladino[3]

[1] MFEE department, EDF R&D, Chatou, France

[2]DIPNN - Direction Technique, EDF, Lyon, France

[3]Paul Scherrer Institut (PSI), 5232 Villigen, Switzerland

1. ABSTRACT

Passive safety systems in nuclear reactors rely on gravity as driving force for transferring heat from the core or a hot region into a large reservoir thanks to natural circulation. They are designed to function without the intervention of the operators for several hours or days. Given the large scale of the current SMR designs, the Rayleigh number is expected to be in the order of 10^15. Currently, there are very few experiments that have reached such a high number, so new experiments are necessary to validate thermal hydraulics codes used to predict such behaviour in an accident scenario. The paper shows preliminary calculations using Computational Fluid Dynamics (CFD) to help sizing a new experimental set up at the PANDA facility (PSI, Switzerland) within the OECD/NEA PANDA project.

2. INTRODUCTION

A nuclear renaissance is currently underway due to the need for cleaner, cheaper and sovereign energy. Small Modular Reactors (SMRs) are an economically attractive proposition due to the cost savings obtained through the modularisation of components [1]. Another opportunity for cost savings comes from simplifying reactor systems. The passive safety systems are designed to operate using gravity as the driving mechanism for heat removal in case of an unintended shut-down of the reactor. Generally, they should be simple enough to allow free flow circulation and should operate during large periods of time, normally up to several days.

Some SMR designs rely on having the containment building inside a large reservoir or pool of water to provide passive cooling. In the case of an accident, the interior of the containment will be filled with high temperature steam which will, in turn, transfer the heat to the outside pool through the containment wall. Inside the pool, the heated fluid will move upwards along the heated wall, and a recirculation loop will start. In time, the pool will increase in temperature and stratification can be present. The characteristic non-dimensional number for the heat transfer along a heated wall is the Rayleigh number defined as:

$$Ra = \frac{\rho\beta g \Delta T H^3}{\alpha\mu} \quad (1)$$

Where ρ is the density, β is the thermal expansion coefficient, g is the gravity, ΔT is the temperature difference between the wall and the bulk, H is the height of the containment, α is the diffusivity and μ is the kinematic viscosity.

*Corresponding Author: juan.uribe-torres@edf.fr

In the SMR designs, the large geometrical scales and high temperature differences can lead to values of Ra in the order of 10^15 to 10^16. Experimental studies for heated wall in pools are available [2] but generally at lower Ra. At the high Ra present in the SMRs the data are very limited, mainly because of the large-scale facility needed to obtain them. Numerical studies have been done [3, 4] but there is still a need for high quality experimental data for validation of thermal hydraulics codes and methods.

This paper presents the preliminary calculations done using Computational Fluid Dynamics for sizing a new experimental campaign at the PANDA facility at PSI [5].

3. METHODOLOGY

The PANDA facility has been used for various thermal hydraulics tests. It has a modular structure with six pressure vessels allowing for different configurations. For the natural convection experiments, a vessel (called "Vessel 1") of 4 m in diameter and 8m in height is available. The vessel can be filled with water at a pressure up to 10 bar. The current proposition for the design is to insert a cylindrical vessel representing the reactor containment building in the pool. This vessel, called "Containment Simulator" (CS) can be filled with high pressure high temperature steam to provide the necessary heating to the pool. The result is a natural convection loop in the pool that removes heat through the CS wall. It is therefore necessary to dimension the CS and set the operating conditions to be able to obtain a Ra number of the order of 10^15.

A 2D axisymmetric CFD model has been created to investigate the heat transfer loop and perform parametric studies. The code used is EDF open-source software *Code_Saturne* (see https://www.code-saturne.org). With the present model, the influence of varying dimensions, operating conditions, mesh refinements and numerical parameters can be studied.

The domain is shown in figure 1 which includes conjugate heat transfer through the solid part of the CS (solid thickness = 0.01m). The boundary condition at the inside of the CS is set to an exchange coefficient based on the wall temperature, the saturation temperature of the steam and the liquid film Reynolds number using the experimental correlations . The other walls are set to adiabatic conditions since Vessel 1 is insulated. The free surface of the pool is set to a zero-flux condition for all variables. The momentum and temperature equations are solved iteratively using an unsteady algorithm using the SST turbulence model and the Elliptic Blending Reynolds Stress Model (EBRSM). Various mesh resolutions have been tested until mesh independency was achieved with the finest mesh having about million control volumes. The near-wall resolution was such that the non-dimensional distance, y+ and non-dimensional temperature, T+ were less than one at the interface between solid and fluid (the Prandtl number in the present configuration is of the order of 2).

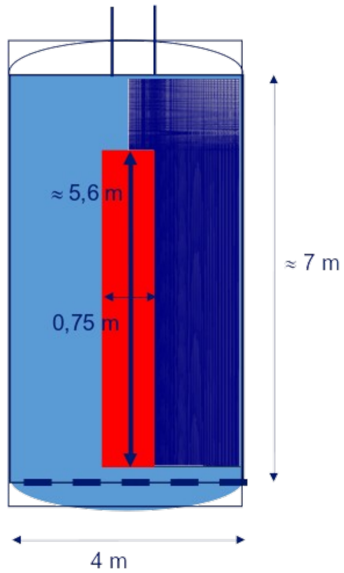

Figure 1: Computational domain and dimensions.

The operating conditions were set within the limits of the facility capacities, namely T_{sat}=175 and P=9 bar inside the CS. The initial temperature in the pool was set to 40 °C. The first 20 minutes were simulated.

4. RESULTS

Figure 2a shows the Nusselt number (Nu=hL/k, with h the heat transfer coefficient, L the characteristic length and k the thermal conductivity) as a function of the Ra number for the two converged turbulence models. The dotted lines show the experimental correlations for laminar and turbulent boundary layers as shown in [3]. Figure 2b shows the wall temperature as function of the height. It can be seen that the maximum Ra achieved at the top of the CS is Ra ≈ 10^15. There is a large difference in the prediction of wall temperature between the two models which highlights the need for good quality experimental data for validation. The decreasing temperature observed with the SST model doesn't seem to be physical as it is shown in existing experiments [6] .

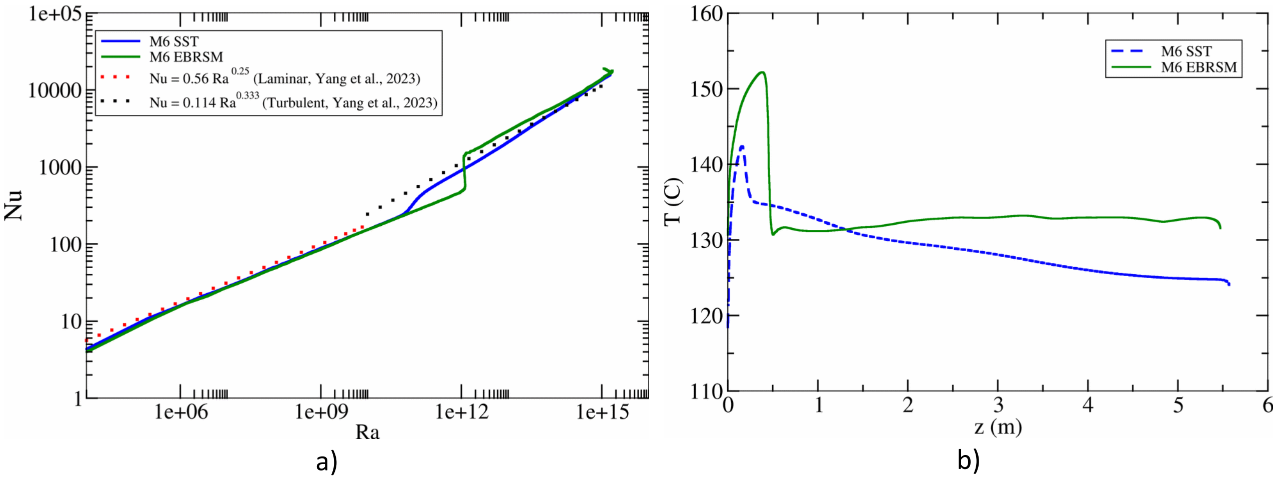

Figure 2. Nusselt number (a) and wall temperature (b)

5. CONCLUSIONS

Initial calculations suggest that it is possible to obtain Ra ≈ 10^15 within the physical and operational constraints in one of the PANDA vessels. By pressurising the water pool, the properties of water change allowing to reach the desired Ra at a lower height compared to ambient pressure tests. Most of the numerical tests show only small sensitivity to varying parameters in the near wall region (Nu, Ra). The exception being the choice of turbulence model, which has a large impact in the prediction of the near wall temperature gradients and therefore heat transfer coefficient. The sizing analysis has shown that is possible to reduce the containment simulator diameter to save energy during testing. The simulations can be used to set up the temperature measurements very close to the wall that are needed to capture the boundary layer. Preliminary computations show that transition seems to occur z < 0.5m in a region of ≈ 20cm, although none of the models tested so far are designed to model transition correctly.

REFERENCES

[1] THE NEA SMALL MODULAR REACTOR DASHBOARD, NEA No. 7650, © OECD 2023

[2] Vliet, G. C., Liu, C. K., An Experimental Study of Turbulent Natural Convection Boundary Layers. *Journal of Heat Transfer*. 1969

[3] Yang, S., Bieder, U., Studer, E. LES of natural convection along a side-heated vertical wall in a water cavity applied for the scale of Small Modular Reactor (Ra ≈ 10 ^15). *Nuclear Engineering and Design* 414 (2023).

[4] Angelis, A., Reinke, N., Ambrosini, W. Assessing water-wall behaviour for a light-water Small Modular Reactor with the aid of CFD analyses. *Annals of Nuclear Energy* 184 (2023).

[5] Paladino, D. and Dreier, J., PANDA: A Multipurpose Integral Test Facility for LWR Safety Investigations. *Science and Technology of Nuclear Installations*. (2012).

[6] Freitag, M., Schmidt, E. W., von Laufenberg, B., Gupta, S. Experiments on Passive Heat Removal of Immersed Containments by Laminar and Turbulent Convection at Very High Rayleigh Numbers. NURETH-20, Washington, D.C. (2023).