BJRS

BRAZILIAN JOURNAL
OF
RADIATION SCIENCES
10-03A (2022) 01-14

Simulation of steady-state and transient loss of cooling accident of a channel in a reactor plate-type fuel element

Monteiro[a], D.B.; Maiorino[b], J.R.

[a] *Universidade Nove de Julho / Departamento de Exatas, CEP: 03155-000, São Paulo, SP, Brazil*

e-mail: deiglys.monteiro@gmail.com

[b] *Programa de Pós-graduação em Energia / Universidade Federal do ABC /Centro de Engenharia Modelagem e Ciências Sociais Aplicadas (CECS), CEP: 09210-580, São Paulo, SP, Brazil*

e-mail: joserubens.maiorino@ufabc.edu.br

ABSTRACT

The suitable cooling of fuel elements in a nuclear reactor is an important requirement that must be met to avoid that the fuel temperature rises above the safety limits according to the reactor design. During the reactor operation, there are transients that could lead to a temperature that overcomes these limits, such as those related to the cooling system. The CFD codes are tools that could aid in the understand of the phenomenon during such transients, allowing to access details of the flow that are not possible, or are possible only with limitations, by using other kind of codes or experiments. In the present work, the results obtained using ANSYS-CFX® code for the IEA-R1 reactor during a steady state and transient of slow loss of cooling accident event are presented. The results obtained shown a good agreement with experimental data reported and works that used this reactor as case of study. These results are part of a research in which the main objective is to simulate the flow of the coolant in the fuel element channels during transients. These results would support an initial analysis of the flow during the transition from forced to natural convection that occurs when the coolant flow falls below a settled value and the valve on the bottom of the core opens by gravity, aiming to understand better phenomena involved and the limitations of the models available in the ANSYS-CFX® code.

Keywords: **Transients, CFD, Cooling, Phenomena, Reactor.**

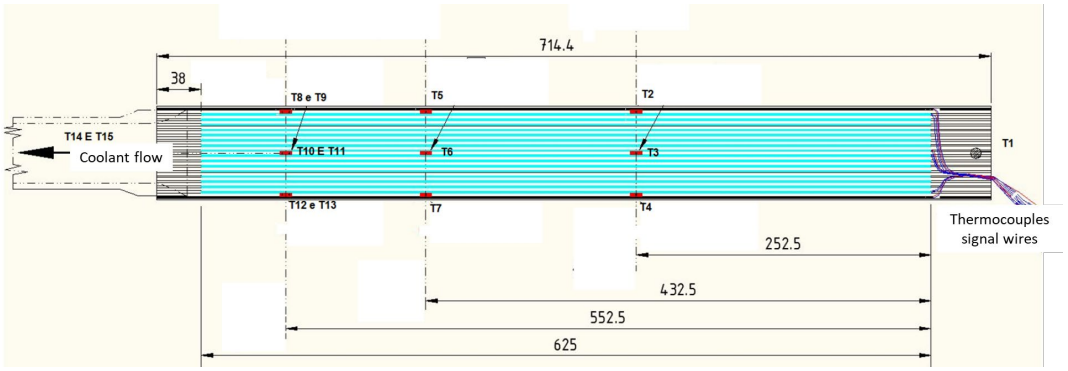
ISSN: 2319-0612
DOI: https://doi.org/10.15392/2319-0612.2022.1829
Submitted: 2022-01-19
Accepted: 2023-01-25

1. INTRODUCTION

The suitable cooling of the fuel elements in a nuclear reactor is an important requirement that should be meet since it avoids that the fuel temperature rise above its design limits, when it could reach a temperature high enough to melt and release the fission products, featuring an accident. Different kinds of fuel elements have also different types of channels through the coolant flows in different patterns, according to the channel geometry and fluid and flow characteristics. In the case of a plate type nuclear reactor, each fuel element has several channels through the coolant flows. The channels consist in the space between two consecutives fuel plates. By flowing through these channels, the coolant removes the heat produced in the fuels and keep them in an adequate temperature [1-5].

The IEA-R1 is a plate type research reactor in which an investigation of the flow parameters that could lead to a such undesirable state in several stead-state or transient operational regimes could be performed in a safe manner. To aid in this investigation, the IEA-R1 has an instrumented fuel element – IFE to which the pressure drop is known, and that is equipped with thermocouples that are used to measure the temperature in different points through the channel length in some channels of the IFE [1-4, 6-9]. Despite of this, the IFE instrumentation does not allow to access the details of the flow pattern, which could be done by using a CFD (Computational Fluid Dynamics) code [3]. Figure 1 show a schematic view of the IFE with the main dimensions and the positions of the thermocouples [1].

Figure 1: *Schematic view of the IFE – main dimensions and thermocouples positions.*



Other relevant features of the IFE are summarized as follows [1, 2, 7, 9]: a total of 15 thermocouples (TCs) are present in different channels. The TCs are encapsulated in small aluminum discs made with the 6061 alloy, avoiding the Galvanic corrosion and possible accidents involving leak of fission products to the reactor pool. The alloy was also chosen since it has a high thermal conductivity. The Al-6061 discs containing the TCs are assembled at their positions by interference fit, aiming to reduce the contact thermal-resistance.

Despite most part of the geometry and other important features of the standard fuel element (SFE) used in the IEA-R1 reactor are similar to the IFE, the IFE and the SFE are essentially different. Some of the most relevant features of the SFE related to the present work are [1, 4, 8]: total length (873mm, shorter than the IFE), total length of the plate (625mm, the same for both), total plate width (67.1mm for both), coolant channel thickness (2.89mm for both), active length (600mm for both), active width (62.6mm for both).

A CFD codes are numerical computational codes in which the fluid domain is discretized in small elements of volume, to which the equations of conservation (mass, momentum and energy) are solved for a wide variety of scenarios and flow regimes, such as for steady-state or transient flow regimes, laminar or turbulent flow pattern, etc. As consequence, it is possible to access with a high detail the flow characteristics, such as recirculation zones, local boiling, etc. In the view of the capacities mentioned, the CFD codes are suitable to evaluate the phenomena occurring in a single or in several coolant channels during transients [3, 5, 12]. In literature, a detailed work conducted by [3] evaluates the flow in the IFE of the IEA-R1 reactor using a commercial CFD code – ANSYS-CFX®. The author addressed and compared the CFD results with the experimental data during a SLOFA (Small Loss of Flow Accident). The SLOFAs are events during which the coolant flow is gradually reduced. It could be driven by a failure in the coolant pump, a valve malfunctioning, electrical shortage, electrical fail in a control component, etc. As consequence of the mass flow reduction, the heat produced in the fuel is not adequately removed and accumulates, rising its temperature. If this rise continues until above the fuel design limits, the fuel melts and release the fission products, giving rise to an accident [3, 8, 9].

The simulations conducted by [3] starts in a moment in which the reactor is scrammed after it has been operated with a power of 3.5MW in a steady-state regime. The author simulated the

SLOFA up to the time after which the convection valve at the bottom of the reactor core opens and the coolant flow changes from forced to natural convection. The main objectives of the author were to evaluate if the coolant flow during a SLOFA transient is enough to ensure that the temperature at the fuel element does not surpasses its design limits, as well as to verify the models that produces better results compared with experimental data using as benchmark the coast down curve of the pump obtained experimentally [1-3]. According to [3], the ANSYS-CFX® code was able to simulate the flow pattern suitably. The results shown a good agreement with the experimental data when the κ-ω turbulence model was selected. Notwithstanding, this model requirements in terms of computational effort and time demonstrated to be not prohibitive and the solution convergence occurred without numerical instabilities [3].

Other models available and tested by [3] does not meet these requirements. In this manner, the DNS (Direct Numerical Simulation), which could simulate and predict with high accuracy the pressure and velocities fluctuations, was not choose since it results a computational effort that is prohibitive, requiring dedicated machines and/or supercomputers/workstations to perform the simulations [3]. Other turbulence models tested and that use is not recommended by the author are: a) Zero Equation, b) EVTE (Eddy Viscosity Transport Equation), c) SSG (Speziale Sarkar Gatski), d) κ-ε, e) SST (Shear Stress Transport), f) DES (Detached Eddy Simulation). As reported by the author, the use of these models was discarded due to: a) unable to predict accordingly the physical behavior of the flow, velocity and temperature fields in steady state and transient regimes; b) the energy equation it is unable to converge, even in steady state regimes; c) elevated requirements to spatial and temporal discretization; d) despite the good results obtained in terms of spatial and temporal convergence, the author has ruled out its use since it requires a wall function to capture accordingly the effects of the boundary layer over the flow, overestimating the temperature next to the wall. Notwithstanding, the κ-ε model does not predict accordingly the flow reversion after the convection valve opening at the bottom of the pool; e) its use resulted in numerical instabilities; f) requires elevated spatial and temporal discretization, resulting in prohibitive computational time.

The present work is part of a research which aims to simulate the IEA-R1 transients using the ANSYS-CFX® CFD code when operating at its nominal power (5MW). Simulations with this power would be conducted as part of future works. In the present work, it was conducted

simulations of SLOFA transients that could be experienced by the IEA-R1 reactor operating at 3.5MW from the steady state operation (10s) until 35s after the reactor being shutdown. This first step of the research aims to corroborate the good agreement of the results reported by [3] and the fitting of temperature curve during the steady state operation, after the pump shutdown and with the reactor operating as well as after the reactor shutdown (decay). The convection valve opening was not simulated initially and is subject of future research. Only a central-typical channel of the IFE was simulated in the present work since it has a more stable and symmetrical flow pattern than the lateral channels. No solid domain was included in the simulations, allowing to discretize better the fluid domain without significantly increase the computational effort.

2. MATERIALS AND METHODS

The present work performed numerical simulations using the commercial CFD code – ANSYS-CFX® academic license. The simulations were performed for ¼ of channel in a 3D flow domain to reduce the quantity of mesh volumes and nodes and, as consequence, the computational effort. This was possible since the channel simulated is symmetrical regarding its thickness and width and was assumed that the flow pattern and temperature are also symmetrical [3, 12]. The fluid domain with its main dimensions is show schematically in Figure 2. In Figure 3 the overall mesh details could be observed, including the prisms layers applied at the fluid channel next to the walls. Other relevant features of the mesh are:

- Number of nodes and volumes: 441,640 and 419,160;
- Size of face elements (see Figure 3): 0.15mm
- Quantity of layers applied (through the length of the channel): 60 layers;
- Quantity of prims layers: 25 layers.

Figure 2: *Schematic view of the fluid domain with the main dimensions.*

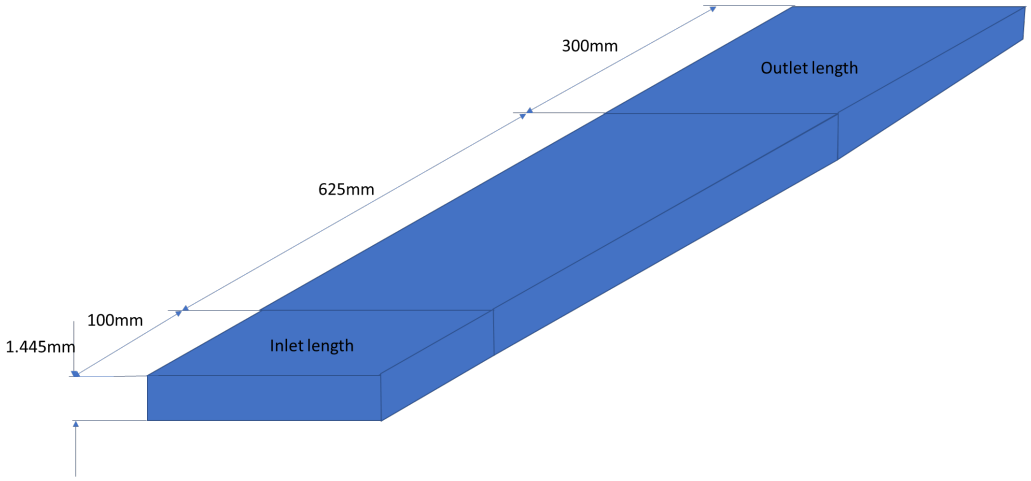
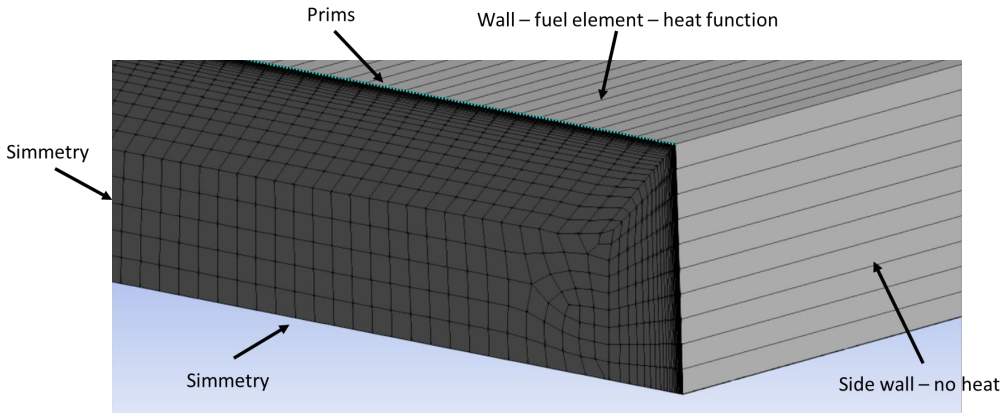


Figure 3: *Mesh view at the channel inlet – prims layers and face elements.*



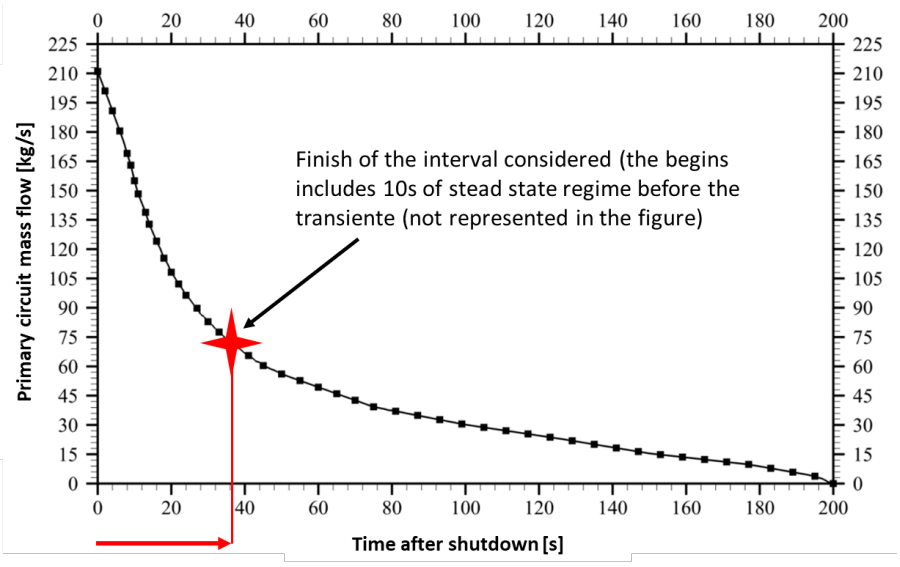
Based on [1, 3, 4, 7, 8], it is possible to assume that the coolant does not experience a bulk phase change (bulk boil) even when the flow is very reduced for a power of 3.5MW, not requiring a more complex approach as necessary for multiphase flows. In this way, it was assumed that only liquid coolant flows through the channel. Since the CFX® code has liquid water (the coolant) in its library, with the properties required to the simulations, no especial care was necessary. It is important emphasize that the same approach was adopted by [3].

As recommended by [3], the turbulence model was setup to κ-ω since this model does not present numerical instabilities that compromise the simulations convergence, as well as treat

properly the boundary layers in thin channels, one of the most important characteristics of the geometry and fluid domain simulated. Similar recommendation is made by [12], so the model was considered adequate for a first approach, attaining for the objectives of the present work. The time step used was of 0.1s. As reported by [3], the difference in the results for smaller time steps are not significant and increases the computational effort. In this manner, this value was considered as adequate for the present work.

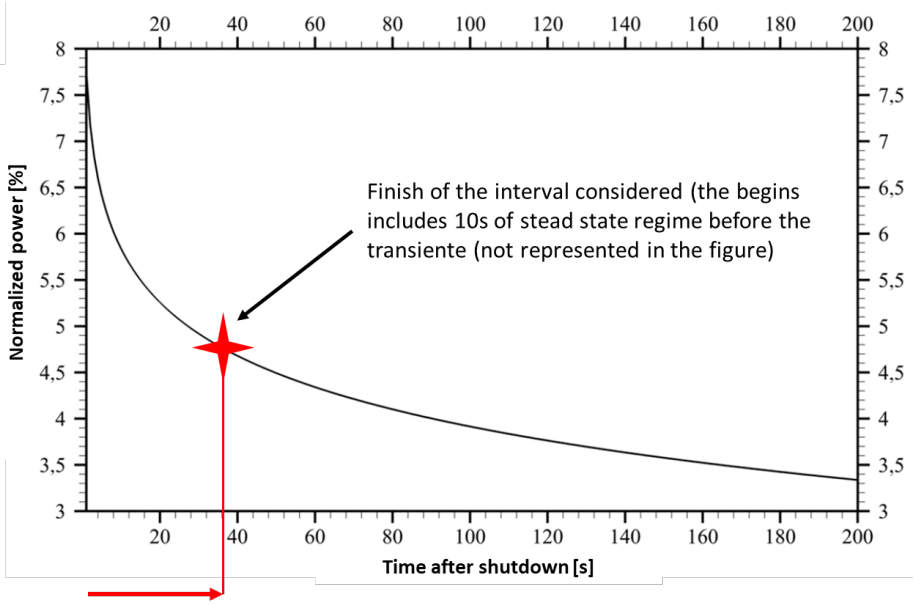
The simulations were performed in the present work including 10 seconds of the reactor steady state operation, after which the coolant pump is shutdown and the mass flow begins to be reduced gradually according to the coast down function, given experimentally by [1]. The 10s of stead state regime was included since the CFD code used had presented instabilities in the beginning of simulation when smaller intervals were tested. The gradual mass flow reduction is due to the flywheel assembled in the pump shaft that stores kinetic energy and releases it to keep the pump running for a time aiming to remove the initial decay heat of the core in case of fails. The pump flow curve is shown in Figure 4. The point indicated refers to the time in which simulation finishes. The stead state operation time interval is not represented. In this manner, a total of 45s was simulated.

Figure 4: *Pump mass flow curve – shutdown curve.*



The IEA-R1 reactor operates during 1.5s after the pump shutdown, being scrammed by the safety system when the flow falls below 90% of its nominal value to avoid damage to the fuel elements [1, 3, 4, 7, 8]. As consequence of the mass flow reduction while the reactor is still operating, the fuel temperature starts to rise. The temperature is reduced after the reactor being scrammed, when the heat produced is due only the decay of the fission products, accounting to about 7.7% of the power in which the reactor was operating just before its shutdown. This decay heat reduces exponentially as the time goes [1, 3, 4]. The decay function used in the present work is the same as given by [3] and which is represented graphically in Figure 5. Similar to the Figure 4, it is indicated in Figure 5 the instant in which the simulation finishes (35s after the pump failure/shutdown). Again, the 10s of stead state operation is not presented, this being included due to the reasons mentioned.

Figure 5: *Heat decay curve.*



The simulation was performed up to 35s after the reactor scram since after this time the general behavior of the flow changes marginally until the convection valve opens and the temperature does not vary significantly. The flow reversion that takes place after the convection valve opening was

not simulated and is part of future research. The initial conditions considered for the simulations in the present work and other relevant data are following detailed [1-4, 7, 8]:

- Initial coolant and fuel plate temperature: 305.7K;
- Mass flow: according to the curve shown in Figure 4. In 0s, the mass flow corresponds to the nominal mass flow, and was used as initial condition. The nominal flow was assumed as constant during the 10s of stead state operation;
- Reactor operating power (before shutdown): assumed as constant and equal to 3.5MW;
- Heat decay power: as given in Figure 5;
- Mass flow correction factors: 1.08 and 0.96 for a central channel and due to the mounting position of the IFE at the reactor core, respectively;
- Coolant properties (regular water): density: 1000kg/m³, specific heat capacity: 1kJ/kgK;
- Other materials: not considered in the simulation. Only the fluid domain was simulated.

3. RESULTS AND DISCUSSION

The results obtained in the simulations are presented in Figures 6 to 9 in terms of the temperature distribution for the TC3, TC6, TC9 and TC14 at IFE. As could be observed, they present the same shape and a good agreement regarding the values reported by [1, 3]].

Figure 6: *Experimental and numerical values for the SLOFA transient –TC3.*

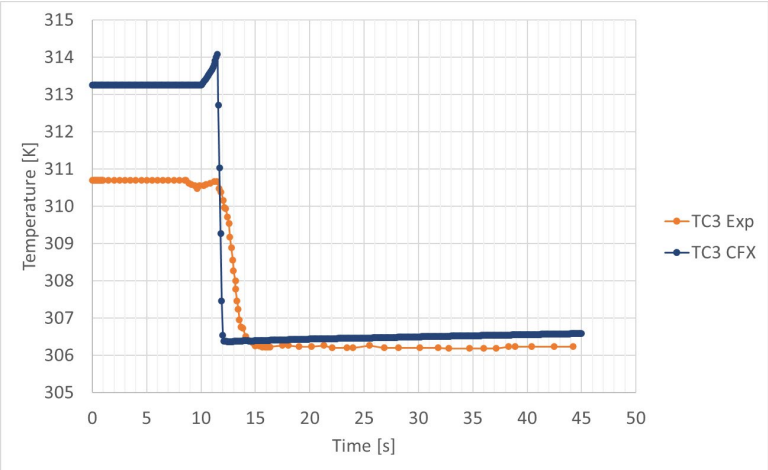


Figure 7: *Experimental and numerical values for the SLOFA transient –TC6.*

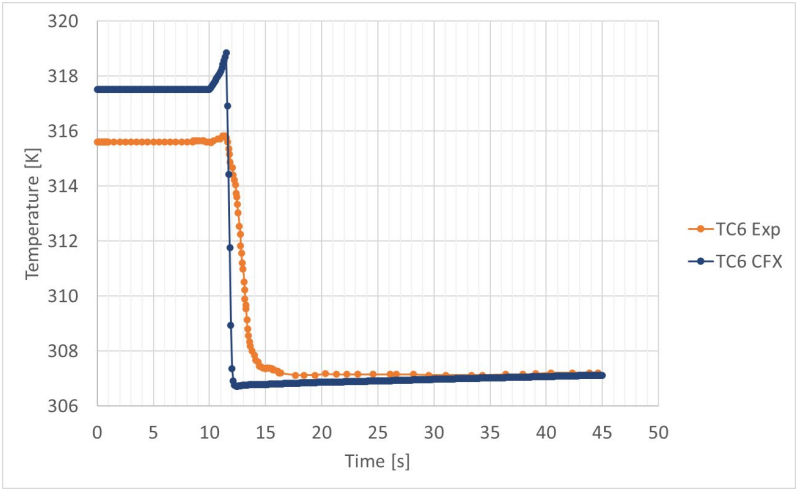


Figure 8: *Experimental and numerical values for the SLOFA transient –TC10.*

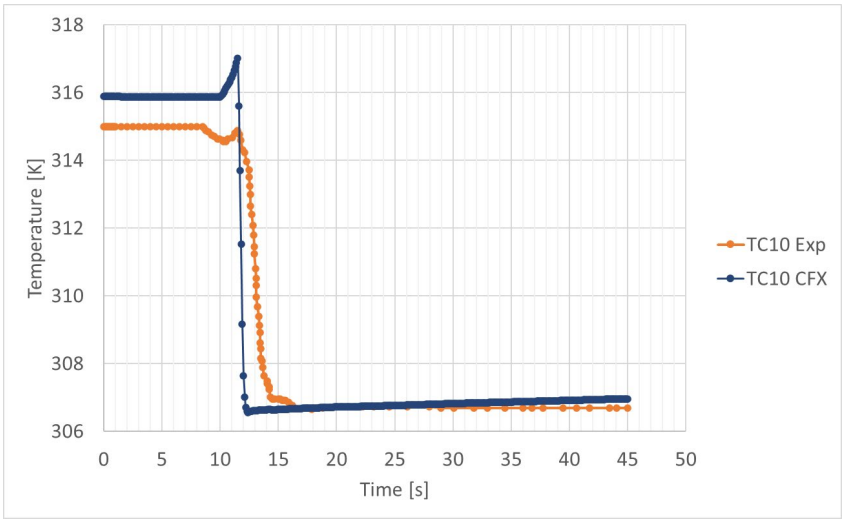
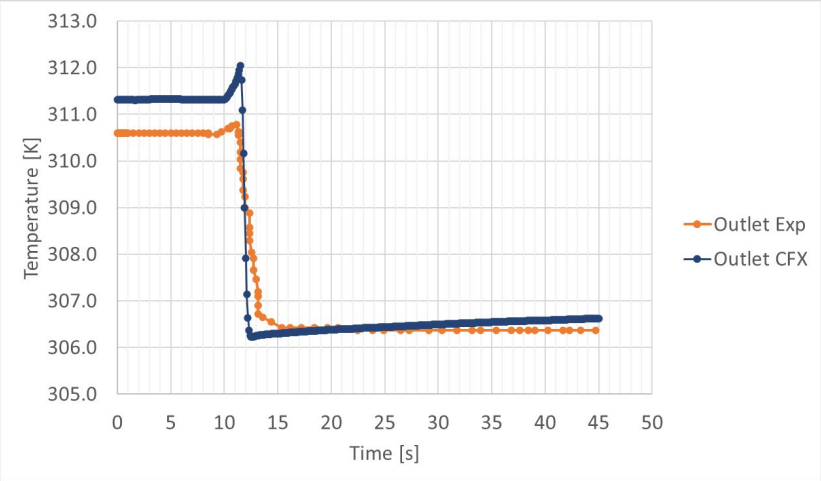


Figure 9: *Experimental and numerical values for the SLOFA transient – TC14.*



As could be observed, the time interval corresponding to the steady-state operation presents the higher difference between the simulated temperature and that experimentally measured to each case, with the maximum difference occurring in the TC3 (~2.5°C), exceeding the margin given by the thermocouple accuracy (±0.5°C). This result agrees well with the reported by [3], which found a difference of 3.1°C. On the other hand, the value found for the TC14 is close to that measured, with a difference of about ~0.5°C. The temperature for the TC14 was not evaluated by [3], so a comparison could not be made. This difference found was attributed to the mesh used and thermal resistance on the contacts of the thermocouples, this last neglected in the present work. In future works, more refined meshes and the thermal resistance would be included and its influence evaluated.

For the TC6 and TC10 the differences found are of about 2.0°C and 1.0°C, respectively, while [3] reports a difference of 2.7°C and 3.4°C, respectively. It could be observed also that the CFX® overpredicts the temperatures in any point of the channel and at the outlet of the IFE. However, this overprediction reduces through the channel length, demonstrating that additional investigation is required to better understand the cause of this divergence and behavior. A possible reason for was attributed to the boundary layer development through the channel and the mesh used to discretize it. Despite the TC3 is positioned almost at the middle of the channel, the flow seems to be disturbed. Another possibility is related to the power distribution/heat generation through the channel. Close to

the TC3 location, the gradient is high while the TC14 is located outside of the active zone of the fuel plate/channel. Finally, it was not considered the real properties of the coolant. In this manner, it is not possible to evaluate their influence over the parameters simulated, requiring more detailed studies in future addressing these features.

Regarding the transient time interval after the coolant pump shutdown, it could be observed the same behavior on the temperatures as reported by [3]: an initial rise followed by a sharp fall. On the other hand, the experiments present a small rise followed by a sharp, but smoother, fall in temperature after the reactor scram. Additional research would be conducted to understand the cause of differences found, initially attributed to the mesh and heat decay function used in the present work.

Due to the sharp fall in the temperatures as predict by the CFX®, the beginning of the plateau after the fall of temperatures begins before the measured experimentally. Initially, this curve shape was attributed to the heat decay function used in the simulations despite the good agreement within the experimental results until 35s after the reactor shutdown. In future, other decay functions would be tested and compared with these initial results.

As a final remark, it should be mentioned that it not expected that the exclusion of the flow reversion of the simulations influences the results obtained or that another turbulence model would result in a different temperature distribution due to the dimensions of the channel and flow characteristics. Despite of it, it is planned to the future additional investigation of mesh refinement, turbulence model influence over the results as mentioned.

4. CONCLUSION

In the view of the results obtained, it could be concluded that the ANSYS-CFX® is a valuable tool in the research and understand of the phenomena involved when nuclear reactors experience a transient such as a SLOFA. Even using a time step and mesh refinement different of that used by [3], the results obtained in the present work are similar, corroborating this author's findings. Additional research is planned for future aiming to evaluate the influence of mesh refinement and heat decay function, as well as the coolant properties.

ACKNOWLEDGMENT

The authors are thankful to Mr. Scuro by share his valuable experience with this reactor, phenomena and the CFD code.

REFERENCES

[1] UMBEHAUN, P.E.; Development of an instrumented fuel element for the IEA-R1 research reactor. **PhD Thesis**, IPEN, Brazil, 2019.

[2] UMBEHAUN, P.E.; Methodology for thermal-hydraulic analysis of pool type research reactors with plate type fuel. **Master Thesis**, IPEN, Brazil, 2000.

[3] SCURO, N. L.; Numerical simulation of a slow loss of coolant accident in a research nuclear reactor. **Master Thesis**, IPEN, Brazil, 2019.

[4] UMBEHAUN, P.E.; TORRES, W.M.; SOUZA, J.A.B.; YAMAGUCHI, M.; SILVA, A.T.; MESQUITA, R.N., SCURO, N.L.; ANDRADE, D.A.; Thermal hydraulic analysis improvement for the IEA-R1 research reactor and fuel assembly design modification. **World Journal of Nuclear Science and Technology**, v.8, 54-69, 2018.

[5] ALMACHI, J.C.; ESPINOZA, V.S.; IMKE, U.; Extension and validation of the SubChanFlow code for the thermos-hydraulic analysis of MTR cores with plate-type fuel assemblies. **Nuclear Engineering and Design**, v.379, 111221, 2021.

[6] SANTOS, M.M.; Structural evaluation of a plate type fuel element for a compact nuclear reactor. **Master Thesis**, IPEN, Brazil, 2019.

[7] DURAZZO, M.; UMBEHAUN, P.E.; TORRES, W.M.; SOUZA, J.A.B.; SILVA, D.G.; ANDRADE, D.A.; Procedures for manufacturing an instrumented nuclear fuel element. **Progress in Nuclear Energy**, v.113, 166-174, 2019.

[8] HAINOUN, A.; DOVAL, A.; UMBEHAUN, P.; CHARZIDAKIS, S.; GHAZI, N.; PARK, S.; MLADIN, M.; SHOKR, A.; International benchmark study of advanced thermal hydraulic safety analysis codes against measurements on IEA-R1 research reactor. **Nuclear Engineering and Design**, v.280, 233-250, 2014.

[9] MAPRELIAN, E.; TORRES, W.M.; JUNIOR, A.B.; UMBEHAUN, P.E.; BERRETTA, J.R.; SABUNDJIAN, G.; Total and partial loss of coolant experiments in an instrumented fuel assembly of IEA-R1 research reactor. **Nuclear Engineering and Design**, v.363, 110610, 2020.

[10] NISHIYAMA, P.J.B.O.; Neutronic and Thermal-hydraulic evaluation of a device for irradiation of a LEU target of UAlx-Al for production of ^{99}Mo in the IEA-R1 reactor. **Master Thesis**, IPEN, Brazil, 2012.

[11] DOMINGOS, D.B.; Neutronic and Thermal-hydraulic evaluation of devices for irradiation of LEU targets of UAlx-Al and U-Ni for production of ^{99}Mo in the reactors IEA-R1 and RMB. **PhD Thesis**, IPEN, Brazil, 2014.

[12] ANSYS User Guide Manual. CFX. Release 20.1. Canonsburg, PA, 2020.