2021 International Nuclear Atlantic Conference – INAC 2021
Virtual meeting, Brazil, November 29 – December 2, 2021

ISBN: 978-65-594-1256-3

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

Deiglys B. Monteiro[1], José R. Maiorino[2]

[1] *deiglysbmonteiro@gmail.com, Departamento de Exatas, Universidade Nove de Julho-UNINOVE, Av. Professor Luiz Ignácio Anhaia Mello, 1363 - Vila Prudente, São Paulo - SP, 03155-000,*
[2] *joserubens.maiorino@ufabc.edu.br, Programa de pós-graduação em Energia (PPGENE)*
Centro de Engenharia e Ciências Sociais Aplicadas (CECS)
Pró-reitoria de Pesquisa (PROPES)
Universidade Federal do ABC
Av. dos Estados, 5001 – Bangú – Santo André, SP, Brazil

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 could flow with different patterns. In the case of a plate type nuclear reactor, each fuel element has several channels to the coolant flow, these formed between two neighboring fuel plates. By flowing through these channels, the coolant could remove the heat produce in the fuel and keep them in an adequate temperature [1, 2].

The IEA-R1 reactor is a research reactor of this kind (plate type) 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 to which the pressure drop is known, and which is equipped with thermocouples that are used to measure the temperature in some of its channels and in different points through the channel length. Despite of this, the instrumented fuel element does not allow to access the details of the flow pattern, which could be done by using a CFD (Computational Fluid Dynamics) code. The CFD code is a kind of code in which it the fluid is discretized in small volumes and the conservation equations (mass, momentum and energy) are solved for a steady-state or transient flow regime in a laminar or turbulent flow pattern. As result, it is possible to access with a high detail level the flow characteristics such as recirculation and turbulence, occurrence of boil, enthalpy change through the channel, among others [1, 2].

In the view of the capacities mentioned, the CFD codes are suitable to evaluate the phenomena occurring in several or in a single coolant channel during transients. In literature, a previous work was conducted to evaluate the flow using a commercial CFD code – ANSYS-CFX® by [2]. In this work, a SLOFA (Slow Loss Of Coolant Accident) transient was simulated and had the results compared with the experimental data. During the SLOFA the coolant flow is gradually reduced, this could be driven by a failure in the coolant pump, break of a pipe, a bad valve functioning, etc. As consequence of the mass flow reduction, the heat produced in the fuel tends to accumulate and its temperature could increase above its design limits, giving rise to an accident [2].

Scuro (2019) had conducted the simulations from the moment in which the reactor is scrammed after it has been operated with a power of 3.5MW and in a steady-state regime to a time after the convection

valve at the bottom of the reactor pool opens, when the coolant flow change and the heat passes to be transferred from forced convection to a natural convection. In that work, the author had as objective to obtain the pump curve, evaluate the fuel temperature during the SLOFA transient and the models that produces better results. According to the author, the ANSYS-CFX® was able to simulate suitable the flow pattern, having the results a good accuracy regarding the experimental values when the κ-ω turbulence model was selected [2].
The present work is part of a research in which the results reported by [1, 2] are been used as reference/benchmark and which have as main objective to perform simulations of the IEA-R1 reactor during transients such as the SLOFA when operating in its nominal power (5MW) using the ANSYS-CFX® CFD code. Thus, in the present work, the preliminary results obtained are presented for a time of 45s, which includes the steady state operation, coolant pump shutdown and mass flow reduction and heat decay process.

2. Methodology

The present work performed the numerical simulations using a 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.
Based on the experimental data [1] and the results reported by [2], the coolant does not experience a phase change even when the flow is very reduced for a power of 3.5MW, not requiring a more complex approach as required for multiphase flows. In this manner, only liquid fluid flows through the channel. Since the CFX® code have the fluid water (the coolant) in its library with the properties required to the simulations, it was not necessary any especial care regarding the coolant physical properties before the simulations being performed.
The turbulence model was setup to κ-ω as recommended by [2] since this model does not present convergence instabilities and treat better the boundary layers in thin channels as is the characteristic of the geometry and fluid domain simulated in the present work. In the present work, it was used a time step of 0.1s since, as reported by [2], the difference in the results for smaller time steps are not significant and also contributes to reduce the time to obtain the results.
The simulations performed had included 10 seconds of the steady state operation, after which the coolant pump is shut and the flows begins to be reduced gradually according the function given experimentally by [1]. This gradual reduction is due to the flywheel assembled in the pump that keeps it running for time. As reported by [1, 2], the reactor still operates during 1.5s after the pump being shut. It is scrammed by the safety system when the flow falls below 90% of its nominal value to avoid fuel damage. As consequence of this mass flow reduction when the reactor is still operating, the fuel temperature starts to rise. It starts to be reduced after the reactor scram, when the power and heat produced is due only the radiological decay of the fission products, which accounts for about 7.7% of the power in which the reactor was operating. This decay power reduces over time exponentially according to a power law. To comparison purpose, it was considered the same the power law as given by [2]. The simulation was then performed up to 35s after the reactor scram, since after it the general behavior of the flow is the same and the simulation of the convection valve and change in the flow, as performed by [2] would be conducted only in future.

3. Results and Discussion

The results obtained demonstrated a good agreement with the experimental values, as show in Figure 1 for the central channel in which thermocouple TC6 is fixed. The same behavior is observed in for the other thermocouples at the channel.

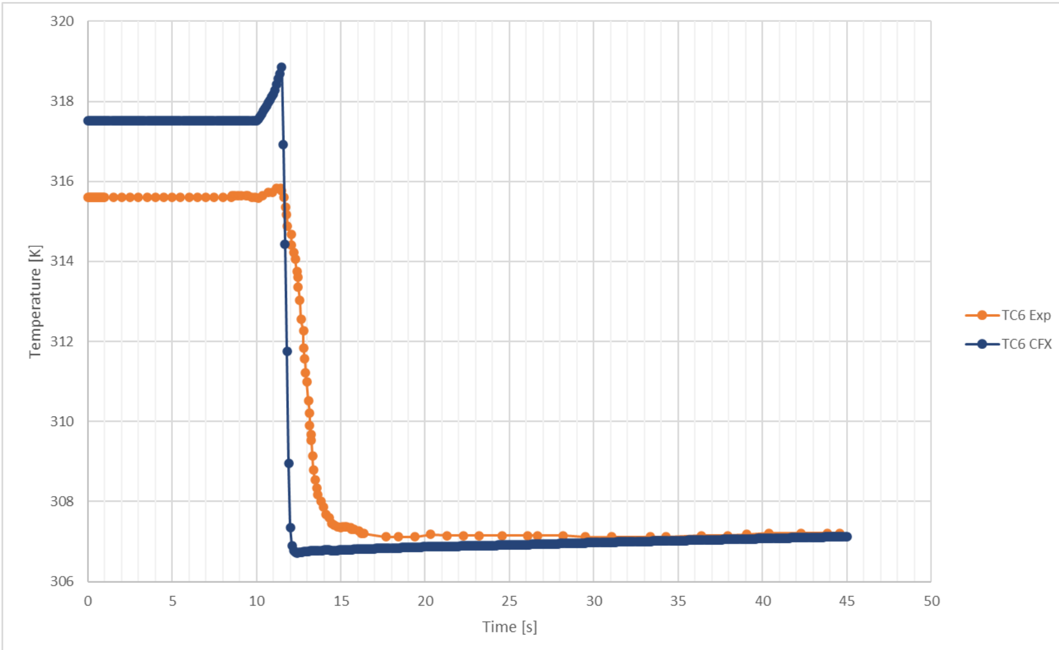

Figure 1: Experimental and numerical values for the SLOFA transient when the flow still is forced.

As could be observed, the steady-state time presents the major difference between the simulated and the experimentally measured, superior to the thermocouples accuracy (±0.5°C), as expected and reported by [2]. Despite of it, the CFD code was able to simulate the phenomena and additional simulations would be performed in future aiming to reduce this difference.

Similarly, it could be observed that for the transient period after the pump being shut and the flow start to be reduced, just before the reactor being scrammed and just after it, the CFX® was able to simulate suitable this transient, despite of the sharp shape of the curve obtained and that some values present a higher error while others have a smaller error. The sharp of the curve obtained with the CFX® is attributed to the equation used to represent the decay process. In a future work, it would be investigated which equation best represents this process aiming to reduce the difference found especially in the time just after the reactor scram. As could be observed, for a longer time the difference found between the experimental and the simulated reduces.

Considering the results obtained, it could be concluded that the code could well reproduce the experimental results reported by [1], being a valuable tool in the understand effort of the phenomena that occurs in a coolant channel during transients experienced by a nuclear reactor. The results of this work also reinforce the results previously obtained by [2]. For future works, it would be investigated the influence of the time step used for the transient part of the simulations, mesh refinement as well as turbulence models and the heat decay equation.

4. Conclusions

In the view of the results obtained, it could be concluded that the ANSYS-CFX® is a valuable tool to study the phenomena during transients that nuclear reactors could experience. Even using a time different mesh and a larger time step, the results obtained reinforces the conclusions reported by [2]. In future, as part of the research under development, the influence of mesh refinement, turbulence models, time step and heat decay equation would be investigated aiming to reduce the difference found between numerical and experimental results. Notwithstanding, as part of future works, the change of the flow experienced after the convection valve opens would be investigated to characterize the full SLOFA transient for the power or 3.5MW before simulations with the nominal power of reactor (5MW) being conducted.

Acknowledgements

The authors are thankful to UFABC for the opportunity to conduct this research as a development of the post-doctoral research developed previously in at the UFABC and to Mr. Scuro by share his valuable experience with this reactor, phenomena and 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] SCURO, N. L.; *Numerical simulation of a slow loss of coolant acidente in a research nuclear reactor.* Master Thesis, IPEN, Brazil, 2019.