doi.org/10.15392/2319-0612.2025.2987
Braz. J. Radiat. Sci., Rio de Janeiro
2025, 13(4) | 01-26 | e2987
Editor: Prof. Dr. Bernardo Maranhão Dantas
Editor: Prof. Dr. Alfredo Lopes Ferreira Filho
Submitted: 2025-07-23
Accepted: 2025-10-29

Thermal-Hydraulic Analyzes of a Slow Loss of Flow Accident in the IEA-R1 nuclear reactor using RELAP and CFD Codes

Costa[a*,b], F. C.; Belchior Jr[a], A.; Torres[a], W. M.; Umbehaun[a], P. E.; Andrade[a], D. A. D.

[a] Energy and Nuclear Research Institute - IPEN, 05508-000, São Paulo, São Paulo, Brazil.
[b] Naval Systems Design Center (Brazilian Navy), 23826-640, Itaguaí, Rio de Janeiro, Brazil.

*Correspondence: costafranklin@yahoo.com.br

Abstract: Among the most critical accidents for the IEA-R1, there is the Loss of Flow Accident (LOFA) in which a sudden and abrupt stop of the primary pump causes the loss of flow. Traditionally, this kind of accident is analyzed by thermal-hydraulic system codes. However, they can overestimate the fluid and fuel temperature along the transient by up to 20%. Moreover, thermal-hydraulic system codes can face difficulties to capture three-dimension phenomenon, such as the natural convection. Meanwhile, Computational Fluid Dynamics (CFD) analysis has shown good results when analyzing open-pool research reactor accidents even dealing with flow inversion. This work presents a thermal-hydraulic analysis of a Slow Loss of Flow Accident (SLOFA) in the IEA-R1 using the RELAP5 code and the commercial CFD code Ansys CFX®. The objective is to combine the advantages of both approaches. The system code was used to find the transient boundary conditions for a CFD model. The CFD software solved the detailed flow pattern in a quarter of a fuel channel. The numerical results showed good agreement with the benchmark data. The peak temperatures were overestimated in only 1.8 °C in the fluid and 3 °C in the cladding.

Keywords: Thermal-Hydraulics, CFD, RELAP, IEA-R1.

doi.org/10.15392/2319-0612.2025.2987
Braz. J. Radiat. Sci., Rio de Janeiro
2025, 13(4) | 01-26 | e2987
Editor: Prof. Dr. Bernardo Maranhão Dantas
Editor: Prof. Dr. Alfredo Lopes Ferreira Filho
Submitted: 2025-07-23
Accepted: 2025-10-29

Análise Termo-Hidráulica de um Acidente de Perda Lenta de Vazão no Reator Nuclear IEA-R1 utilizando RELAP e CFD

Resumo: Entre os acidentes mais críticos para o reator IEA-R1, destaca-se o Acidente de Perda de Vazão (*LOFA*) no qual uma parada abrupta e repentina da bomba do primário causa a falta de vazão. Tradicionalmente, este tipo de acidente é analisado por códigos termo-hidráulicos de sistemas. Entretanto, este tipo de código pode superestimar a temperatura do fluido e do combustível durante o acidente em até 20%. Mais ainda, os códigos termo-hidráulicos de sistemas não são capazes de captar fenômenos tridimensionais. Por outro lado, a análise de Dinâmica dos Fluidos Computacional (*CFD*) têm apresentado bons resultados em análises de acidentes de reatores de piscina mesmo em situações com inversão térmica de escoamento. Este trabalho apresenta uma análise termo-hidráulica de um Acidente de Perda Lenta de Vazão (*SLOFA*) no reator IEA-R1 usando o código RELAP5 e o código de *CFD Ansys CFX*®. O objetivo foi combinar as vantagens dos dois tipos de códigos. O código de sistemas foi usado para encontrar as condições de contorno para um modelo de *CFD*. O código de *CFD* resolveu detalhadamente o escoamento em um quarto de canal combustível. Os resultados numéricos mostraram boa concordância comparado com os dados de *benchmark*. Os picos de temperatura foram superestimados em aproximadamente 1,8 °C no fluido e 3,0 °C no revestimento.

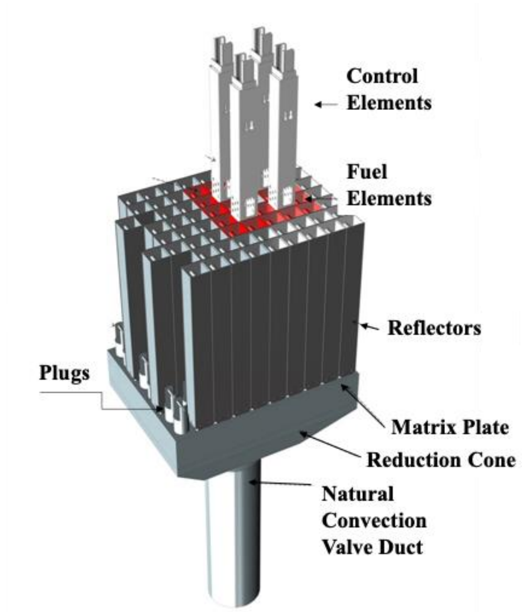
Palavras-chave: Termo-hidráulica, *CFD*, *RELAP5*, IEA-R1.

1. INTRODUCTION

During the licensing process of nuclear power plants, a qualitative risk analysis is conducted to identify the possible accidents that may impact the reactor safety. The most limiting accidents are then postulated and deeply studied quantitatively to predict their consequences. Thus, the dose rates can be verified, as well as the release of radioactive material during accidents.

Within this context, a safety study [1] was conducted to increase the IEA-R1 thermal power from 3.5 MW to 5 MW. The IEA-R1 (**Figure 1**) is an open-pool research reactor located at the Energy and Nuclear Research Institute (IPEN) within the São Paulo University Campus. Its main design parameters and operating characteristics are presented in Table 1. Among more than 60 initiating events analyzed, four were considered the most limiting accidents and studied quantitatively. One of these is the Loss of Flow Accident, which may result from a failure such as a break in the primary pump axis.

Figure 1: IEA-R1 reactor core



Source: [2].

Table 1: RELAP5 results in steady state condition (3.5 MW)

Parameter	Value
Thermal Power	5 MW
Enrichment	<19,75 %
Fuel	U_3Si_2-Al (3.0 gU/cm^3)
Cladding	Al
Maximum core inlet temperature	40 °C
Mass Flow in a Fuel Element	22,8 m^3/h
Cladding Maximum Temperature	95 °C
Active fuel height	600 mm

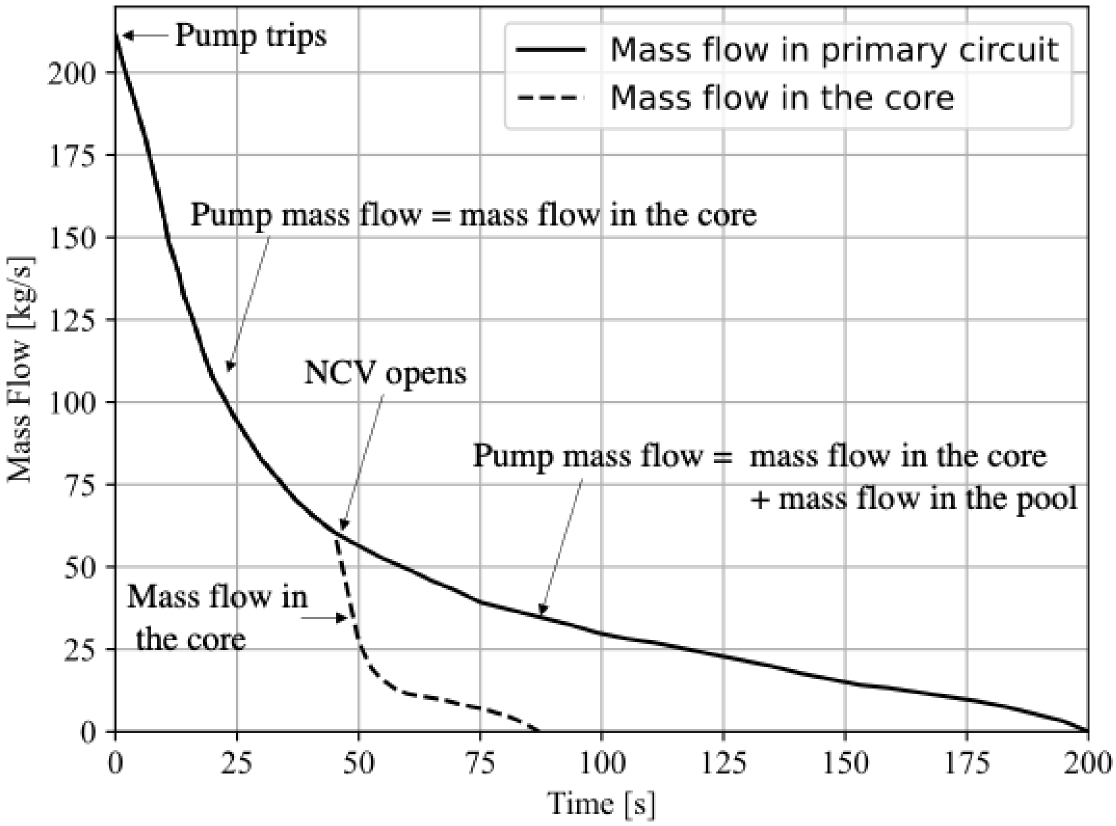
Source: [3].

Despite the occurrence of such accidents, the IEA-R1 is considered quite safe, since it is cooled by natural convection even under accidental conditions. This is possible due to its Natural Convection Valve (NCV), which connects the core outlet to the primary circuit. During normal operation, the pressure in the core is lower than that in the pool, which keeps the valve connected to the core. The valve is not mechanically locked between the core and the primary circuit. When the flow in the core decreases, due to an accident or a normal shutdown, the pressure difference between the core and the pool is no longer sufficient to keep the valve connected, and it starts to fall by gravity. When this occurs, the primary pump draws water from the core and the pool, as illustrated in the

Figure 2. Consequently, it is mandatory to determine the core flow decay in order to calculate the fluid, fuel and cladding temperatures during the transient.

To address this need, the quantitative safety analyses are traditionally performed using numerical codes developed and qualified for this purpose. Among these codes we can refer to RELAP5 and CATHARE codes. These tools can model the entire plant behavior during transients and accidents, and they are known as thermal-hydraulic system codes. As a result of such analysis, some safety parameters can be verified, for example, fuel maximum temperature, fluid temperature and Departure from Nucleate Boiling (DNB).

Figure 2: Mass flow during a SLOFA transient



Source: adapted from [3].

The system codes model the fluid flow in a unidimensional form. Specifically, the RELAP5 software models a plant with volumes and junctions. In this case, the equations for mass, energy and momentum for liquid and vapour phases are solved by the code [4]. Additionally, it solves two equations for non-condensable gases and soluble boron. The finite differences method is used. The nuclear plant models are made of built-in basic components, such as pipe, single volumes, time dependent volume and junctions, single and multiple junctions, turbine and pump.

A unidimensional simulation can present certain limitations when modelling some physical phenomena such as the natural convection. According to the literature [5], the thermal-hydraulic system codes have restrictions to predict low-velocities flows, thus, they are able to foresee the natural convection phenomenon, but with limitations due to its tridimensional nature. These limitations are not a safety concern, since these codes tend to overestimate the safety parameters. In a benchmark study [6], five independent research teams modeled a SLOFA in the IEA-R1 reactor with RELAP5, CATHARE, MARSAT and PARET. The objective of the study was to compare the simulation results from four softwares against experimental data obtained with an IFA (Instrumented Fuel Assembly). As a conclusion, the results showed differences up to 20% when comparing the maximum cladding temperature against the experimental results. More recently, a RELAP5 model [7] was developed to study the SLOFA in the IEA-R1 reactor. This model overestimated the maximum cladding temperature by 8.7%.

Both works ([6] and [7]) did not capture the flow decay in the core accurately when the NCV was released. This was probably due to difficulties in modelling the associated area and pressure losses in that region. As a result, the flow decayed more abruptly than observed in the experiment after NCV decoupling. Consequently, the flow direction changed from downward to upward early during the transient when compared with the benchmark. This behavior also helps explain the differences in the peak of temperatures reached in the fuel and cladding. Reference [7] identified three main reasons for this problem:

1) Less thermal inertia from the computacional model;
2) Pressure loss coefficients greater than the real core pressure losses, and,
3) Possible non-linearity in the valve velocity when it falls, taking more time then the considered for the simulation (2 seconds).

On the other hand, Computational Fluid Dynamics (CFD) codes have been widely used in the industry and have proven its efficiency to treat tridimensional problems. Therefore, these codes are of huge importance when studying natural convection in nuclear reactors. However, in the nuclear industry, the safety analysis demands tested and validated codes. Althought CFD codes have been extensively used in nuclear industry and its acceptance by the nuclear community have grown, there are some challenges for its use specially in the two-phase flow field [8].

A recent work [9] investigated the SLOFA transient in the IEA-R1 reactor to assess the performance of the Ansys CFX® [10] in simulating nuclear transients. The results showed that the software accurately predicted the flow inversion during the natural convection phase. The cladding maximum temperature was overestimated by 7.3%. with this peak occurring at the same time as in the experimental transient. These findings corroborate that CFD analyses can provide valuable contributions to reactor safety assessments.

In addition, several CFD analyses have been performed to study accidents in open-pool reactors. A Fast Loss of Flow Accident (FLOFA) in the International Atomic Energy Agency (IAEA) 10 MW generic Material Testing Reactor (MTR) was analyzed using the CFD tool Ansys Fluent [11]. The results showed good agreement with other qualified software. Another study [12] investigated a Loss of Flow Accident in a 22 MW MTR using CFD. The results were compared with those obtained from the PARET code and demonstrated good accuracy.

However, using only CFD to model the SLOFA transient is not practical, since determining the flow decay in the core is not straightforward with this type of tool. Non-convectional strategies would be required, such as the use of immersed solid or dynamic mesh in the complex geometry of the reactor.

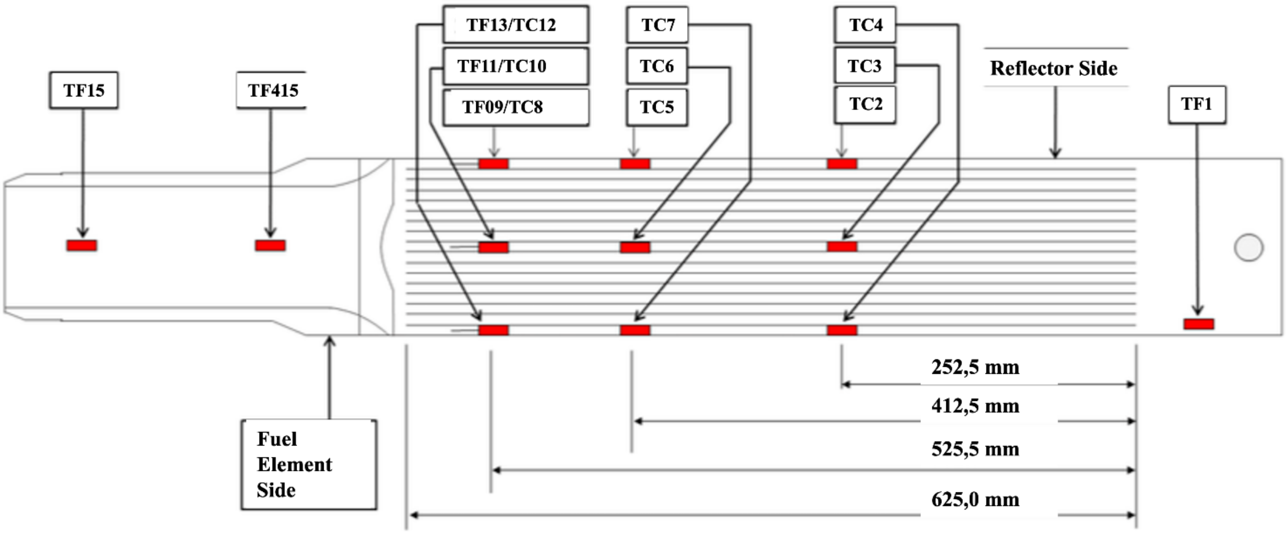
Concerning the combined use of thermal-hydraulic system codes together and CFD tools, some studies have analyzed the advantages of this approach. A potential increase in the thermal power of the Tehran Research Reactor (TRR) was evaluated through a hybrid analysis using RELAP5 and Ansys Fluent [13]. Another research team [14] coupled the RELAP5-3D with a CFD code to study the despressurization of a test facility in a two-phase flow problem. The authors concluded that the coupling strategy has the potential to enhance the accuracy of the safety parameters predictions. In other work [15], the diffusion of low-temperature water in a downcomer during boron dilution transient was investigated. Two methods were compared with the benchmark data of the International Standard Problem No. 43, RELAP5 only and RELAP5/Fluent. The hybrid method (RELAP5/CFD) demonstrated better agreement with the benchmark, which can be attributed to the complex three-dimensional mixing behavior in the downcomer, captured by the CFD code.

The objective of the present work is to analyze the SLOFA transient in the IEA-R1 reactor using RELAP5 code and Ansys CFX®. The RELAP5 determined the boundary conditions during the transient problem, since determining the core flow decay only with CFD tools is not practical. The RELAP5 model developed in this study followed the recommendation from [7] and adopted a different strategy to represent the NCV. Subsequently, a detailed CFD analysis was performed to predict the complex nature of natural convection and the flow inversion during the accident. The ideia for using a hybrid analysis is to take advantage of both softwares and improve the accuracy of the results.

1.1 Benchmark Data

In order to obtain thermal-hydraulics and neutronics benchmark data, a series of tests [3] were conducted in the IEA-R1. During these tests, an Instrumented Fuel Assembly (IFA) was used to evaluate fluid and cladding temperatures. Fifteen thermocouples were installed throughout the fuel element. Twelve inserted inside an aluminum disc positioned between the fuel plates and 3 in the inlet and outlet positions of the fuel, as shown in Figure 3.

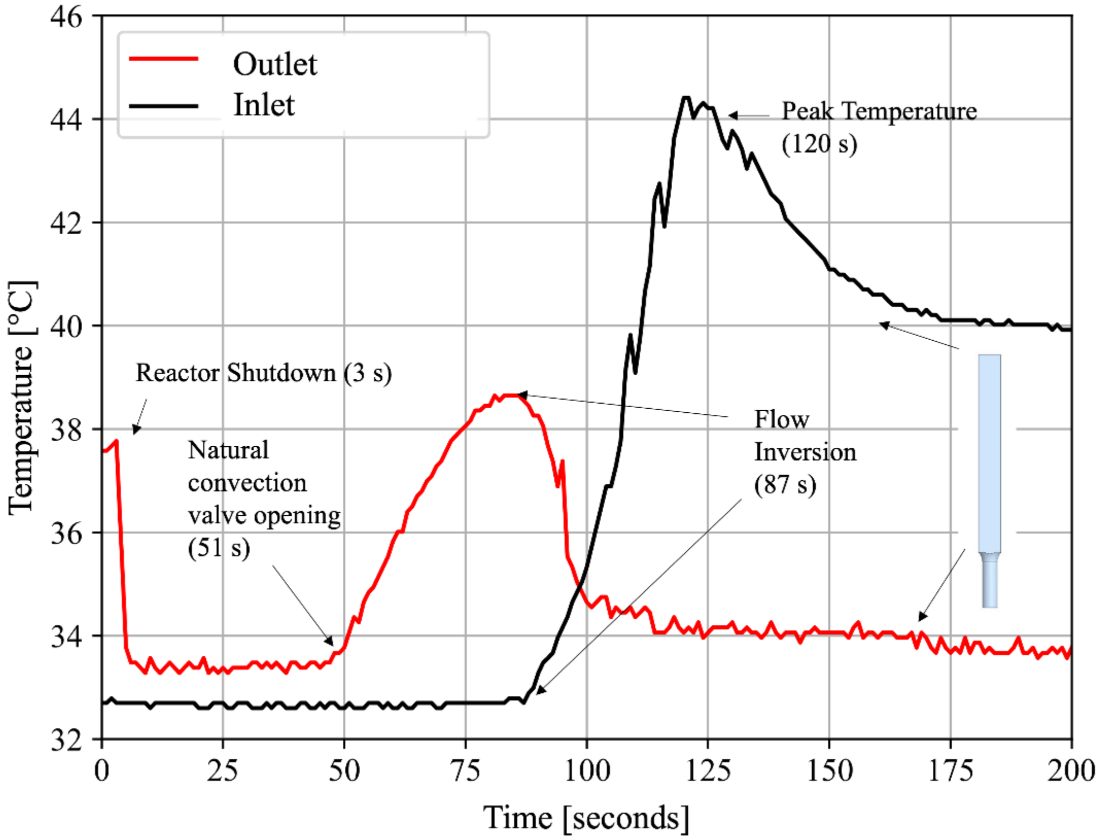
Figure 3: Instrumented Fuel Assembly (IFA)



Source: [9].

A SLOFA was simulated using the IFA. The complete scenario is presented in Figure 4, which shows the experimental result for inlet and outlet thermocouples. During the experiment, the reactor was stabilized in 3.5 MW power. After that, the primary pump was shut down, which corresponds to 0 seconds. When the mass flow rate in the core achieved 90% of its nominal value, the reactor was shut down by low flow (3 s). It can be observed that the outlet temperature in the fuel element increases until reactor shutdown due to flow decrease. After reactor shutdown, the outlet temperature decreases abruptly caused by the fission chain reaction interruption. At about 51 seconds, the natural circulation valve was opened since the pressure difference between the pool and the core was no longer capable of supporting the valve weight. The outlet temperature started to increase again, until 87 seconds, because the core was decoupled from the primary loop. At this time, the fluid temperature increased in such a way that its lower portion started to ascend due to density difference causing the flow inversion. Finally, at 120 seconds the fluid reached its maximum temperature in the upper part of the fuel element. After that, the fluid temperature started to decrease again because of the stabilization of the natural circulation phenomenon.

Figure 4: Fluid temperature during a SLOFA (TF1 in red and TF15 in black)



Source: adapted from [3].

2. MATERIALS AND METHODS

2.1 RELAP Model

The RELAP5/Mod 3.3 was used to calculate the core flow decay during pump coastdown. The reactor core, primary loop circuit, primary pump and the water pool were modeled. The nodalization is shown in Figure 5, where the names in parentheses indicate the type of component used to represent each reactor part.

The reactor core model consists of four channels. The average channel represents 23 fuel elements, and the IFA corresponds to the Instrumented Fuel Assembly. Both channels have a corresponding heat structure with a specified thermal power with the axial power profile. Each heat structure has a layer for the fuel meat and the aluminum cladding. The other channels correspond to the reactor bypass and the irradiation device. The bypass is formed by small gaps in the core which allows coolant flow instead of passing through the fuel elements.

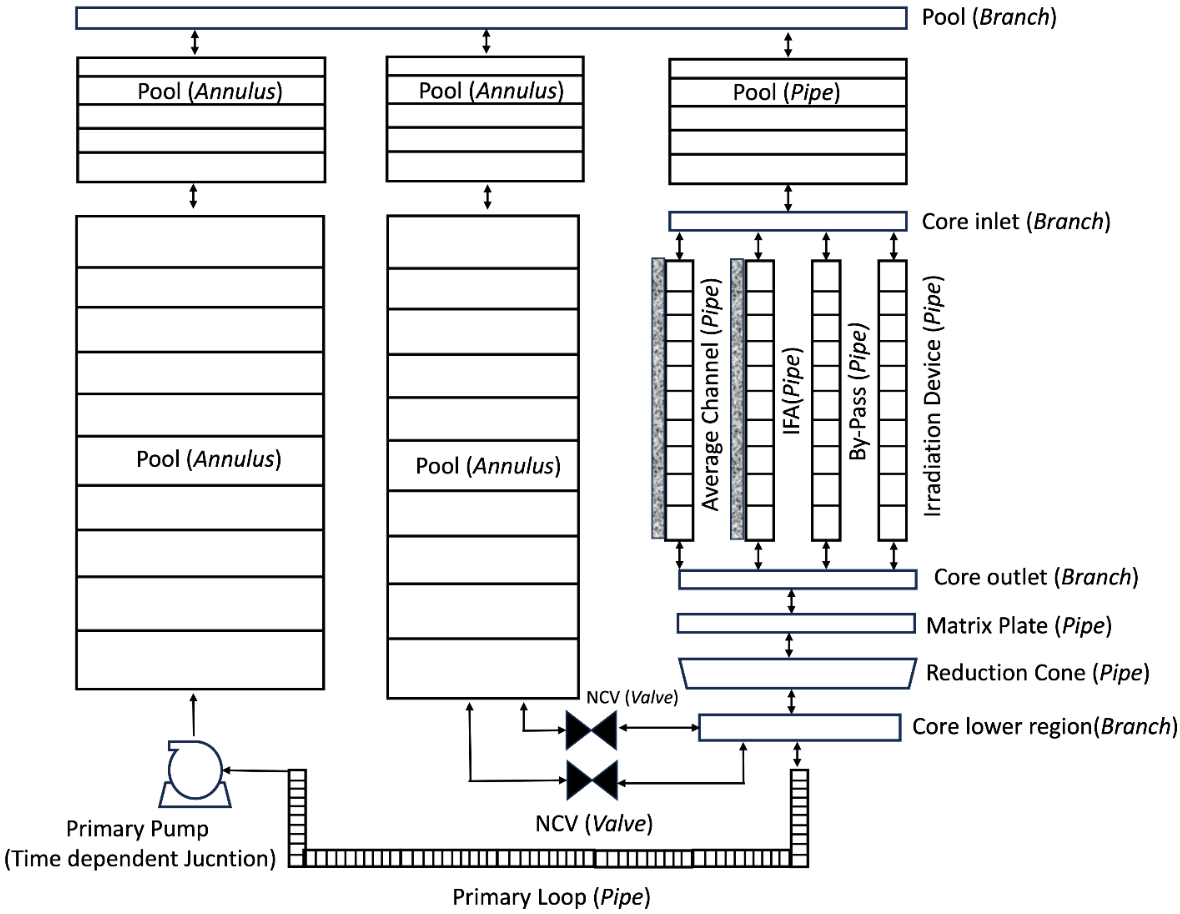
Unfortunately, RELAP5 does not include a dedicated model for the Natural Convection Valve. Therefore, alternative strategies must be studied to represent this specific component and, consequently, improve the safety analysis, since it impacts directly the peak of temperatures.

In this context, regarding the RELAP model of the reactor, the present work adopted a different approach from that proposed in [7] to represent the Natural Convection Valve. In reference [7], the NCV was modeled with two valves: the first opened at the onset of decoupling, and the second opened immediately afterward. The total time to open both valves was 2 seconds. The purpose of using two valves is to adjust the flow area according to flow regime.

In the present article, the idea of using two valves was kept, but with a slight modification. The first valve opens under the same criteria, when the core flow achieves 58 kg/s, within 2 seconds. The second valve opens only when the flow reverses upwards, indicating the onset of the natural convection. The total flow area of the two valves is equivalent in both articles.

The primary pump flow was imposed with a time dependent junction. The reactor is shutdown when the water flow reaches 90% of its nominal value. The pool water was modeled with annulus in the lateral area and one pipe above the core.

Figure 5: IEA-R1 nodalization

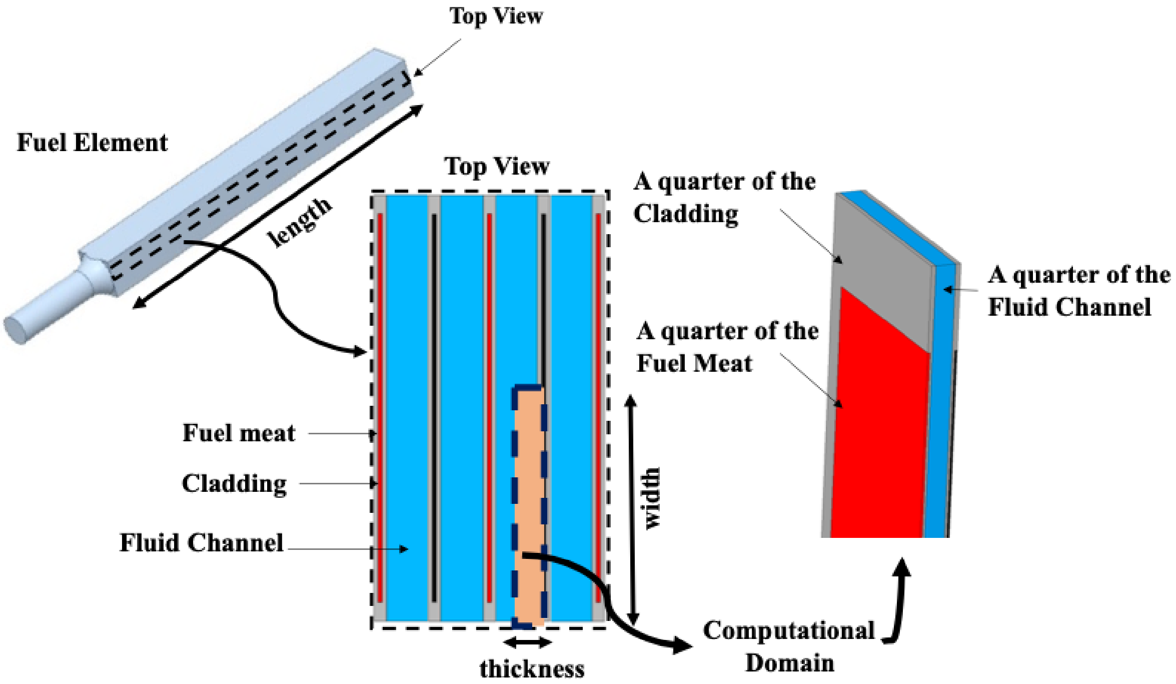


Source: Author.

2.2. CFD Model

The CFD domain was a simplified model considering one quarter of the IFA channel, as shown in Figure 6. In this model, the cladding, fuel meat and fluid channel of a central plate was considered. The dimensions are defined in Table 2.

Figure 6: CFD Computational Domain



Source: Author.

Table 2: Computational Domain Dimensions [mm].

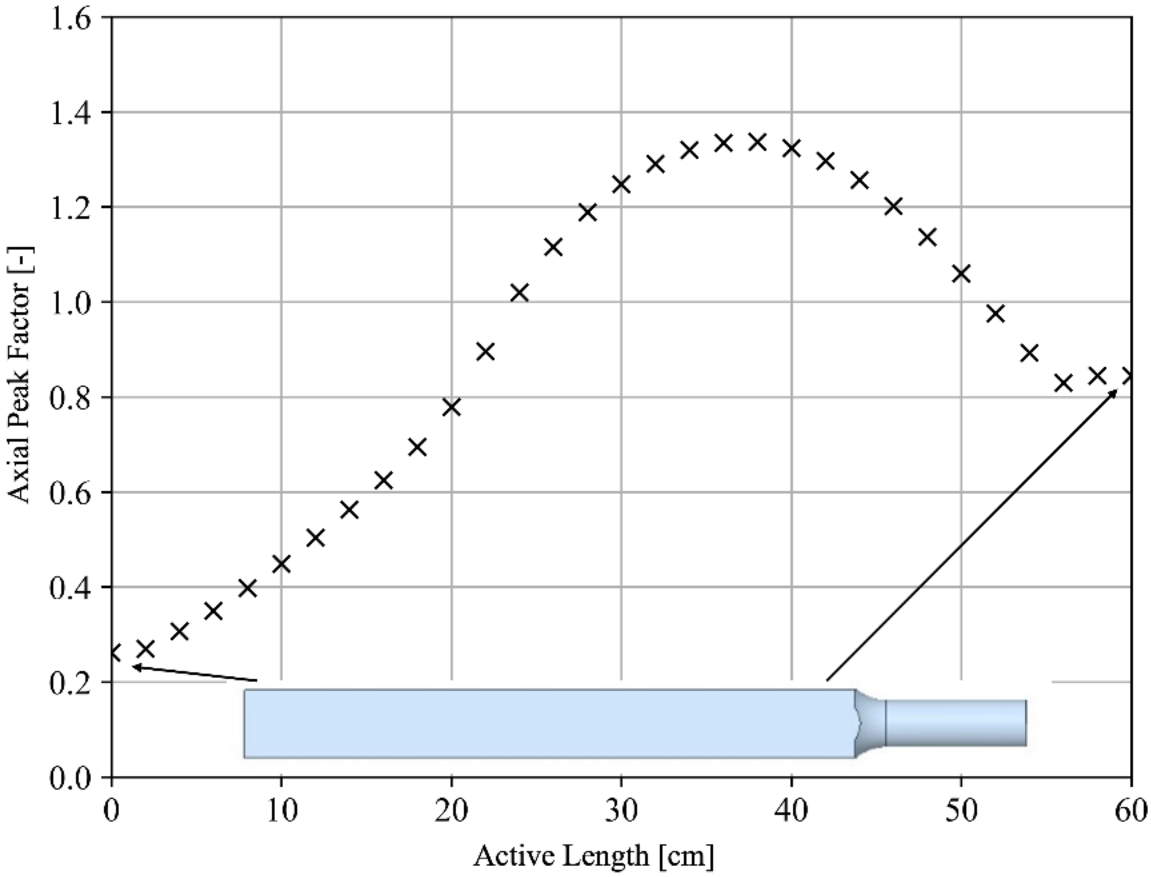
-	Fuel Meat	Cladding	Channel
Thickness	0.760	0.760	1.445
Width	30.175	33.750	33.750
Length	600.00	625.00	625.00

Source: [3].

One steady state simulation was performed to find the reactor conditions before the accident. In this simulation, a power density of 2.47 MW/m^3 for the fuel meat was considered. This value was calculated considering the reactor power (3.5 MW) divided by number of fuel plates in the core (408) and the fuel meat volume. Moreover, the corresponding radial factor for the IFA was considered (0.89). And, to take the axial power distribution for the central plate into account, the curve presented in Figure 7 was introduced in fuel domain. For the fluid domain, it was considered the fluid entering with a mass flow rate equal to 0.090362 kg/s and 32.7 °C. The mass flow rate was calculated considering the mass flow rate of one element which is equal to 22.7 m^3/h. Moreover, this value was reduced by 2% in order to consider experimental results for the central channel [16].

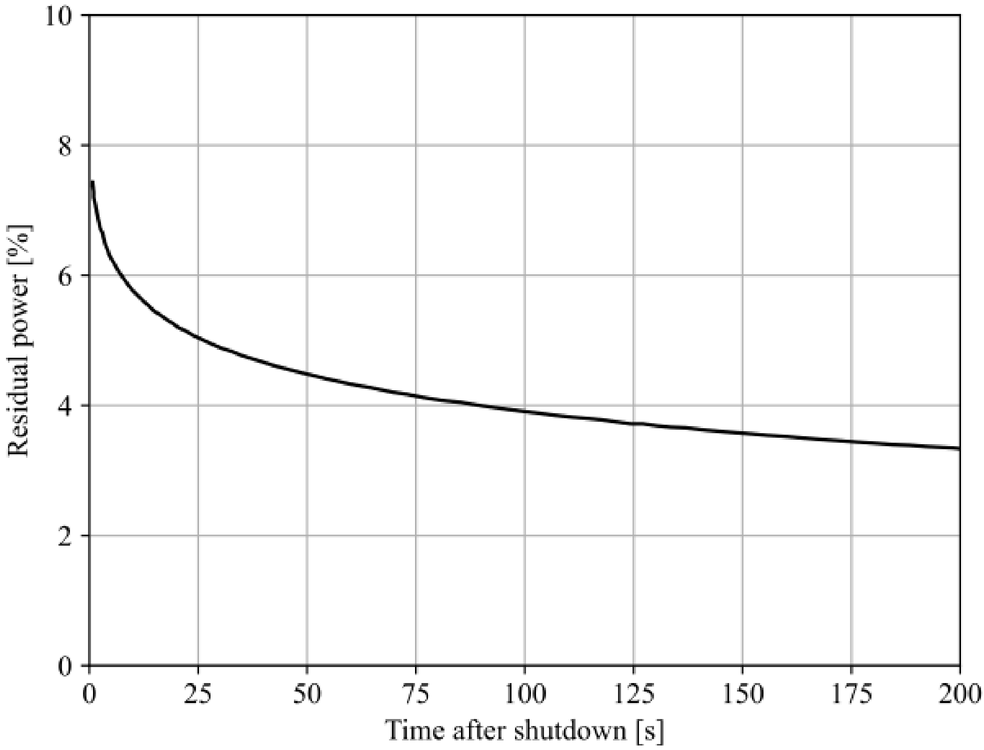
In the transient simulation, the normal power density was considered during the first 3 seconds. After that, the volumetric heat generation was assumed to decrease proportionally according to the heat decay curve (Figure 8). However, the initial volumetric heat generation was increased by 3% to take into account the reactor power history, which was calculated considering the experimental data [3]. The boundary condition for the mass flow rate along the transient was determined with the RELAP5 model.

Figure 7: Axial power distribution for the IFA



Source: adapted from [3].

Figure 8: Residual Power



Source: adapted from [17].

Concerning the mesh parameters, solid and fluid domains were discretized by hexahedral elements, which assures a good quality mesh orthogonality. For the fluid domain, five different meshes were created. These meshes were progressively refined according to the procedure proposed by the literature [18]. The meshes characterisrics is shown in Table 3 and their refinement can be seen in Figure 9.

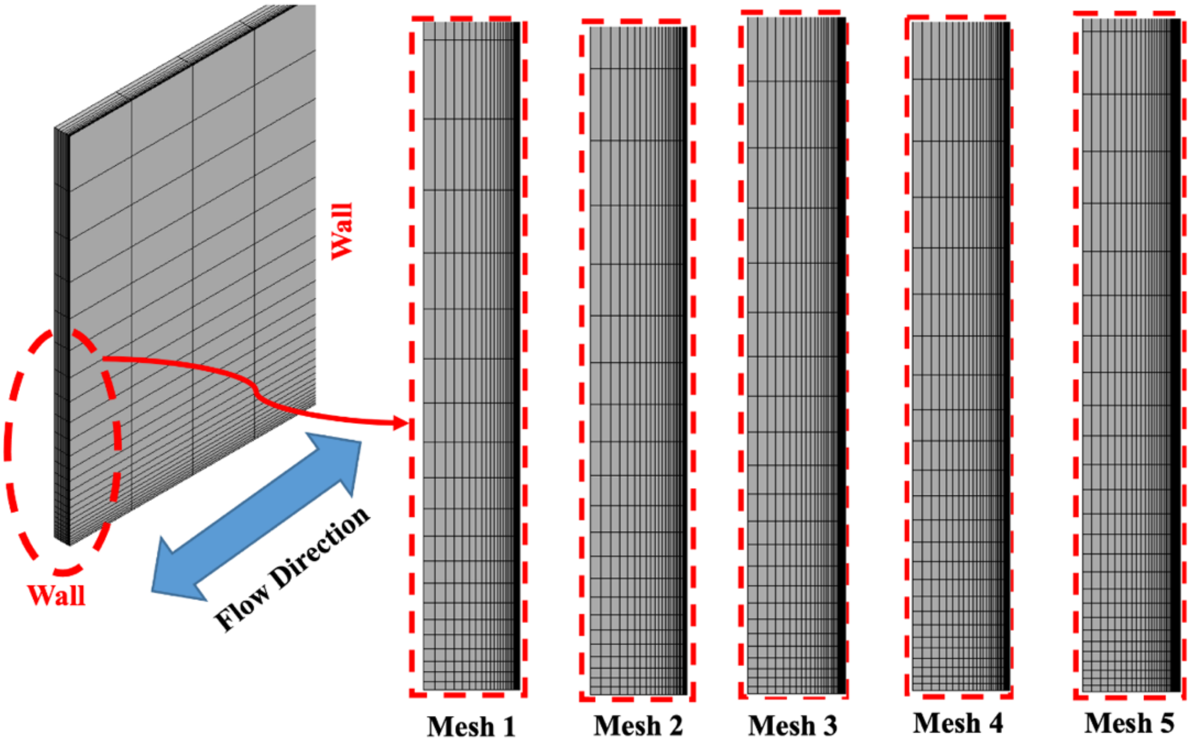
Table 3: Number of elements in fluid domain

Mesh	Thickness	Width	Length	Total
1	28	30	200	247.200
2	30	32	214	284.640
3	32	34	228	328.640
4	34	36	245	387.410
5	36	40	262	447.408

Source: Author.

The mesh convergence was verified considering the steady state case. Figure 10 shows the convergence result for the velocity profile in the outlet boundary, from the plate wall to the middle of the channel. Although there is no significant change in the velocity profile in the steady-state case, Mesh 5 was selected to ensure good mesh quality and to capture physical phenomena in the transient simulation (flow inversion) despite its computational cost. The same occured for the pressure drop, in this case the difference between Mesh 5 and 4 was 0.2%. The time step was determined according to Courant Number. A time step of 0.0001s was used to respect the criteria of Courant number less than the unity.

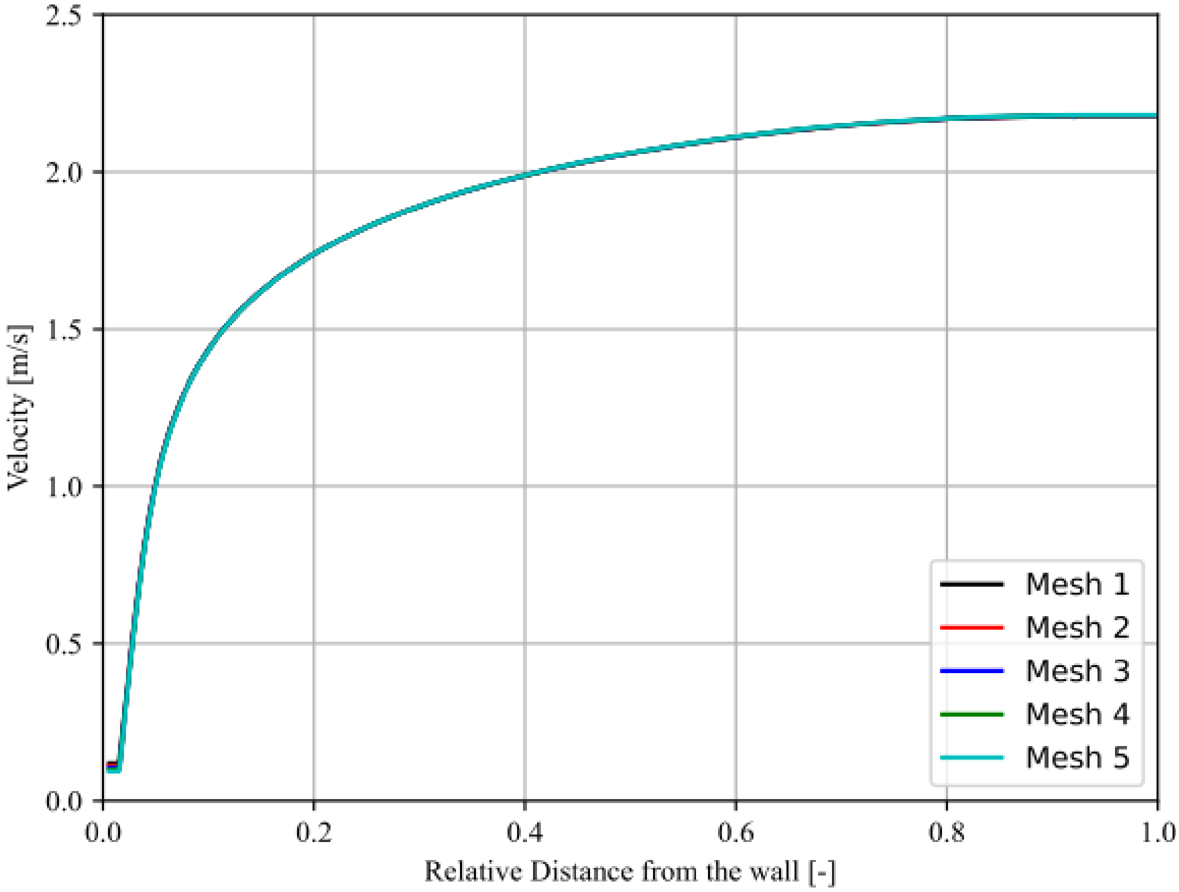
Figure 9: Meshes



Source: Author.

Concerning the turbulence model, the RANS *k*-*ω* was chosen to deal with low Reynolds number flow, which is the case of natural convection. Besides, the Ansys CFX® Manual suggests it for the sake of stability. Moreover, the Automatic Wall Function developed by Ansys specially for this turbulence model was used. This also contributed for convergence stability. Additionally, this turbulence model is appropriate for this kind of problem, since the flow starts turbulent and relaminarize after some time. As the Reynolds number decreases, the k-ω suppresses the turbulent viscosity [10].

Figure 10: Mesh convergence



Source: Author.

The discretization schemes used were the "High Resolution" method for the advective and turbulent terms and a Second Order Euler (backward) method for the transient term.

The Aluminum and U_3Si_2-Al (3.0 Ug/cm^3) properties were considered constant and derived from reference [19] and [20]. Water properties were based on the IAPWS97 [21].

3. RESULTS AND DISCUSSIONS

The RELAP5 results were compared with the theoretical data to determine the error under steady-state condition, Table 4. The results showed good agreement with theoretical value.

Table 4: RELAP5 results in steady state condition (3.5 MW)

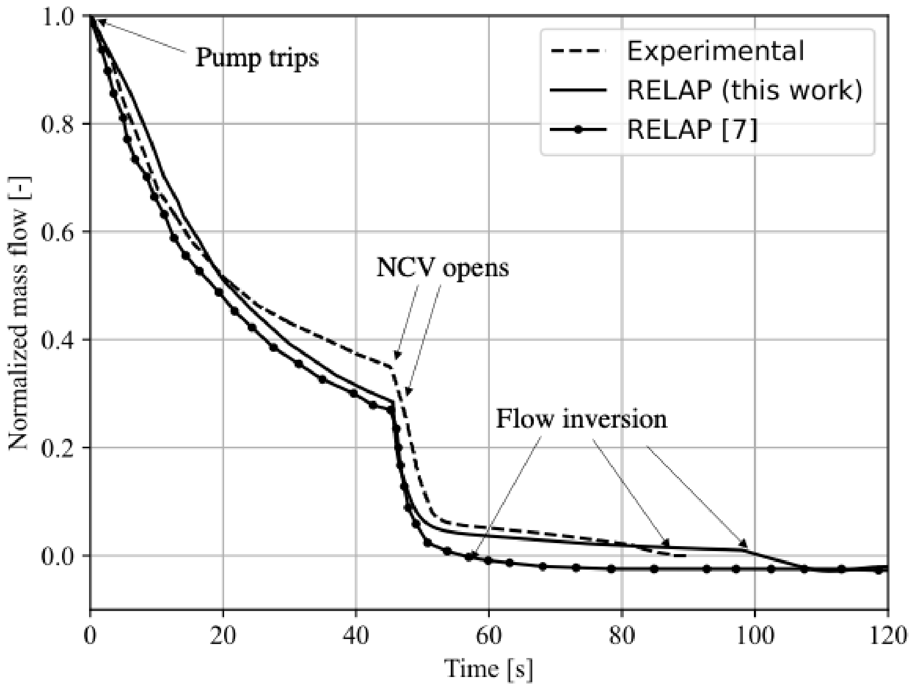
-	RELAP5 Model	IEA-R1	Difference [%]
Thermal Power [MW]	3.460	3.500	-1.130
Mass flow in the fuel element [kg/s]	6.275	6.270	0.079
Temperature Increase in an average channel [°C]	5.497	5.564	-1.210
Pressure drop in a channel [kPa]	3.890	-	-

Source: Author.

The flow decay in the core during pump coastdown is compared in Figure 11. In the RELAP model from reference [7], after NCV opening, the core flow decreases abruptly, reaching 0 kg/s within approximately 10 seconds. Consequently, the flow inversion occurs earlier. In contrast, the model developed in the present work follows the experimental curve more closely after the initial major drop and continues to approach it until the core flow reaches zero. As a result, the flow inversion occurs slightly later.

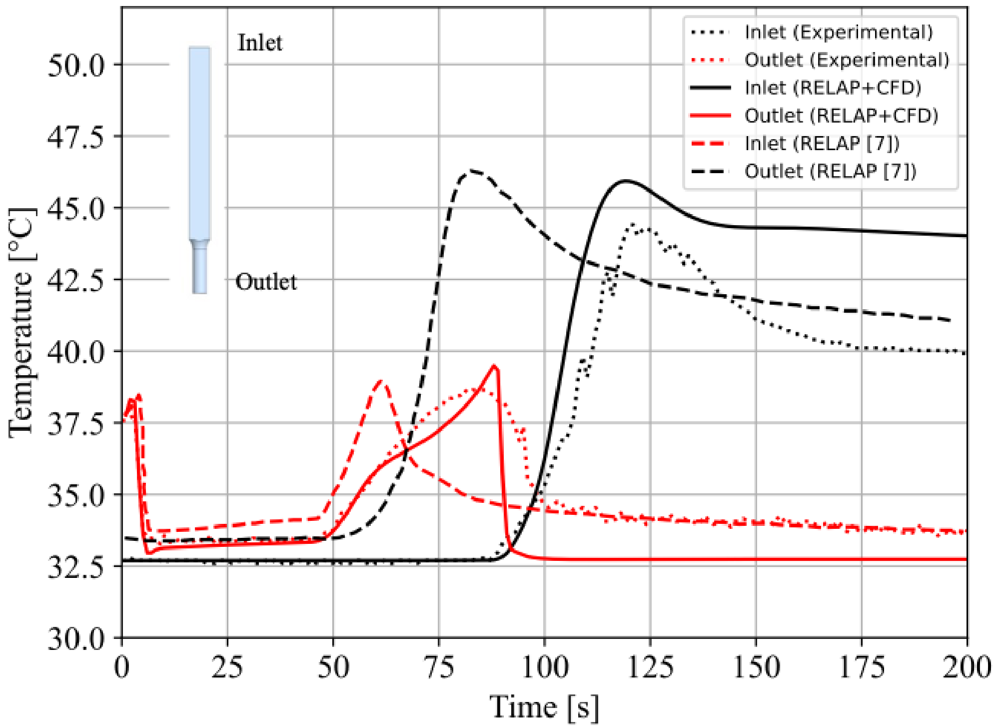
Figure 12 presents the fluid temperature at inlet and outlet. The model developed in this work succesfully captured the flow inversion during the accident. The peak temperature was slightly higher (1.8 °C) when compared with the experiment. The peak temperature occurred practically at the same moment for experimental and numerical results. The RELAP5 model [7] predicted the flow inversion earlier than the experiment, probably due to dificculties in modelling the NCV. The hybrid strategy using RELAP5 and CFD tool appears to provide better accuracy than using only RELAP5 alone.

Figure 11: Core flow decay during the SLOFA transient



Source: Author.

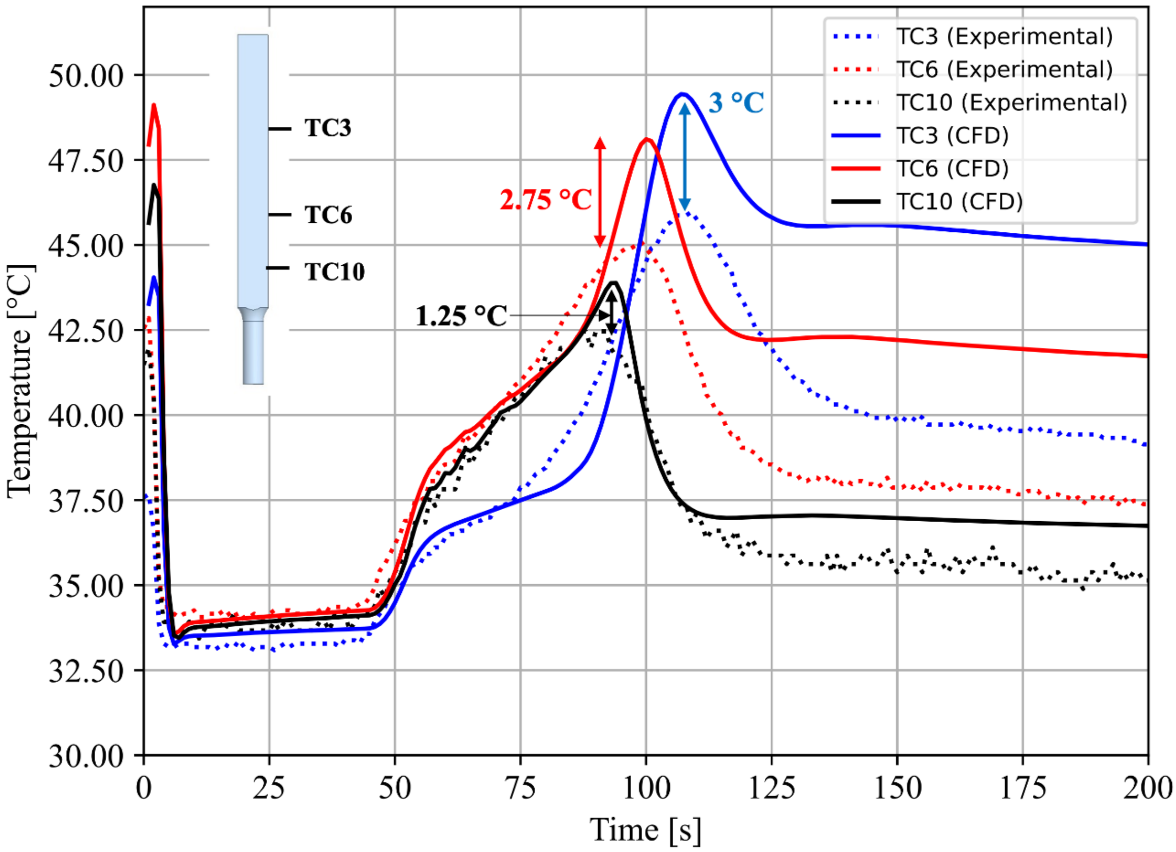
Figure 12: Inlet and outlet fluid temperature during the transient



Source: Author.

The cladding temperature for thermocouples TC3, TC6 and TC10 are shown in Figure 13. Numerical results exhibited good adherence against the experimental ones. The maximum difference between them was 3 °C, in which the error attributed to the thermalcouples is 0.4 °C (measured in umbehaum). In addition, the peak temperature for all thermocouples was basically reached at the same time. Regarding the higher values for numerical results, they can be associated with the thermal resistance between the thermocouples, aluminum discs, and cladding.

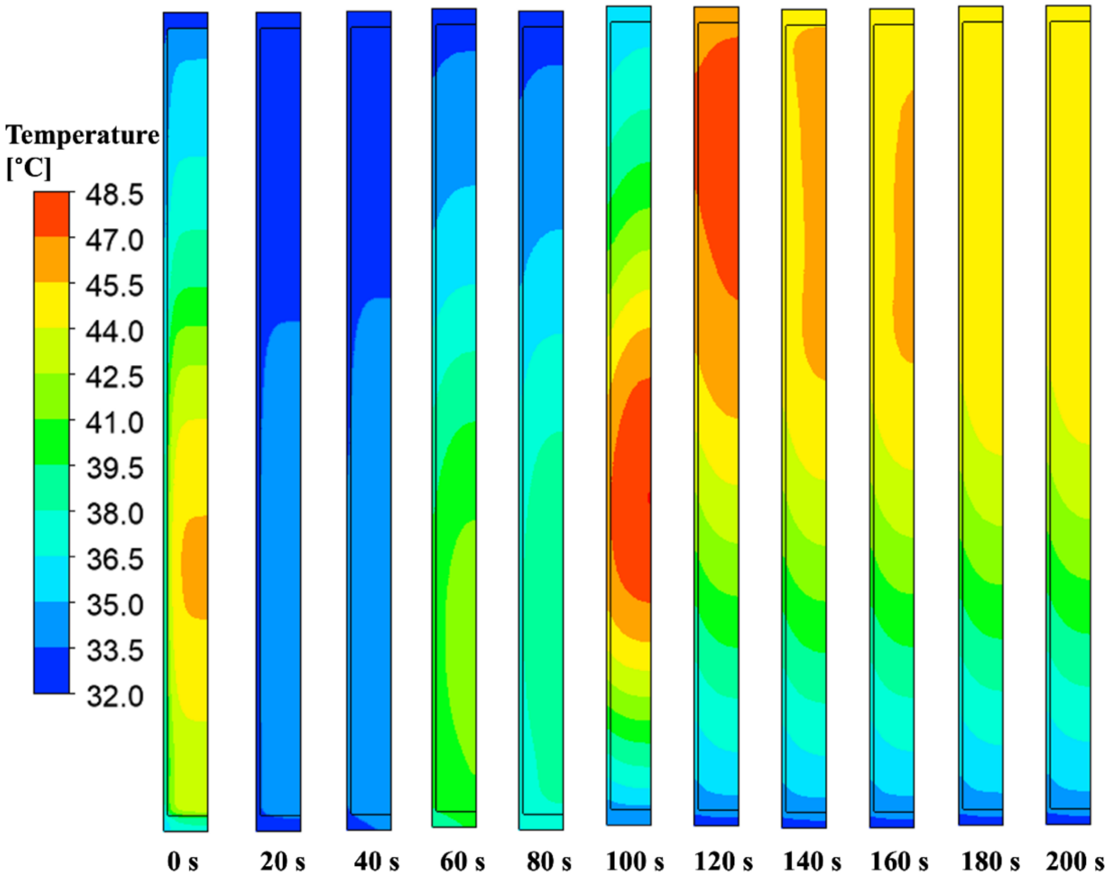
Figure 13: Cladding temperature (thermocouples) (CFD)



Source: Author.

Figure 14 exhibits the cladding surface temperature contour throughout the accident. A peak temperature can be observed between 100 and 120 seconds. Furthermore, a temperature inversion occurred as a result of the flow inversion during the accident. Under normal operation, the highest temperatures are located in the lower part, while after the accident, the upper part exhibits higher temperatures.

Figure 14: Cladding surface temperature (CFD model)

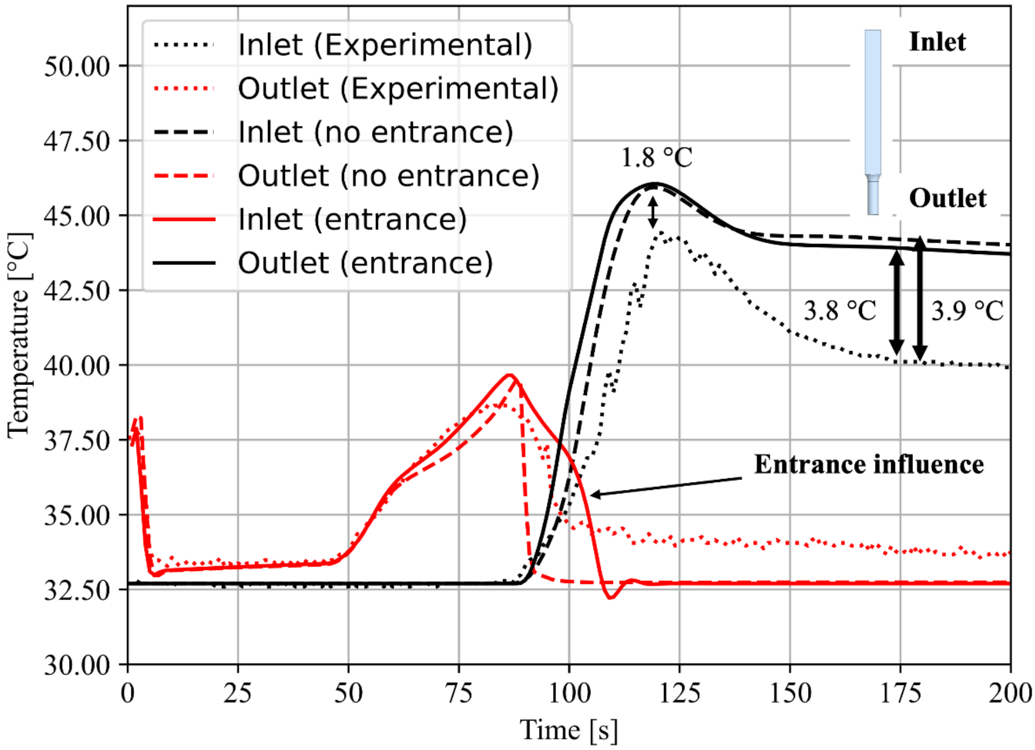


Source: Author.

The influence of the fluid entrance when entering the cone reduction and/or the lower cone of the fuel element could be the reason for the increasing gap between the numerical and experimental results after the peak for fluid and cladding temperature.

Additionally, another domain was used to study the influence of the entrance in the channel of the element over the numerical results. The fluid domain was extended in its upper and lower part of the entire domain (Figure 6). All other parameters were kept the same including the mesh characteristics. The results for all cases are shown in Figure 15. The entrance effect over the results is minimum. Major difference can be observed only after the beginning of flow inversion at about 90 seconds. This happened because the boundary condition at outlet specifies a constant temperature when entering the domain and for the first model there is no entrance.

Figure 15: Fluid temperature (CFD models)



Source: Author.

4. CONCLUSIONS

The IEA-R1 SLOFA transient was modeled using RELAP5 and the commercial CFD software Ansys CFX®. The objective was to evaluate the accuracy of combining the advantages of both softwares and of applying different strategies to represent the Natural Convection Valve of IEA-R1 in RELAP5. Numerical results were compared with experimental data obtained from an instrumented fuel assembly. The simulations showed good agreement. RELAP5 accurately reproduced the core flow decay during the pump coastdown. Regarding the CFD results, in comparison with the benchmark, the maximum deviation in the peak temperature was only 1.8 °C for the fluid and 3 °C for the fuel. In addition, the RELAP5 and the CFD models captured the flow inversion after the reactor shutdown caused by natural convection. These results demonstrate that CFD softwares can greatly contribute to safety analyses. Moreover, there was no significant impact on the fuel element when modelling its entrance.

ACKNOWLEDGMENT

The authors acknowledge Brazilian Navy and IPEN/CNEN for the development of this research work.

FUNDING

The authors declare they received no financial support to carry out this research.

CONFLICT OF INTEREST

All authors declare that they have no conflicts of interest.

REFERENCES

[1] TEIXEIRA e SILVA, A.; MAPRELIAN, A; RODRIGUES, A. C. I.; CABRAL, E. L. L.; MOLNARY, L. D.; MESQUITA, R. N.; MENDONÇA, A. G. "Análise de Segurança do Reator IEA-R1 a 5 MW", In: Congresso Geral de Energia Nuclear, 8.; Encontro Nacional de Física de Reatores e Termo-Hidráulica, 12., 15-20 out, 2000, Rio de Janeiro, RJ. **Anais**, Rio de Janeiro: ABEN, 2020. [Online]. Available in: http://repositorio.ipen.br/handle/123456789/13378.

[2] SCURO, N. L.; ANGELO, G.; ANGELO, E.; PIRO, M. H. A.; UMBEHAN, P. E.; TORRES, W. M.; ANDRADE, D. A. D. "Computational Fluid Dynamics Analysis of an Open-Pool Nuclear Research Reactor Core for Fluid Flow Optimization Using a Channel Box", **Nuclear Science and Engineering**, vol. 197, nº 6, p. 1100–1116, jun. 2023, https://doi.org/10.1080/00295639.2022.2142437.

[3] UMBEHAUM, P. E.; ANDRADE, D. A. D.; TORRES, W. M. Torres; RICCI FILHO, W. "IEA-R1 Nuclear Reactor: Facility Specification and Experimental Results", **International Atomic Energy Agency**, Vienna, Austria, 2015.

[4] SIEFKEN J., L.; CORYELL, E. W.; HAVERGO, E. A.; HORORST, J. K. "RELAP5 Code Manual, Volume I: Code Structure, System Models, And Solution Methods", **U.S. Nuclear Regulatory Commission Nureg/Cr-5535**, Washington, DC, 1995.

[5] D'AURIA, F. Thermal hydraulics in water-cooled nuclear reactors. Oxford: **Woodhead publishing**, 2017.

[6] HAINOUN, A.; DOVAL, A.; UMBEHAUN, P.; CHATZIDAKIS, 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 Design**, vol. 280, p. 233–250, dez. 2014, https://doi.org/10.1016/j.nucengdes.2014.06.041.

[7] CAMPOS, R. C. D.; BELCHIOR JUNIOR, A.; SOARES, H. V.; UMBEHAUN, P. E.; TORRES, W. M.; ANDRADE, D. A. D. "Análise de temperaturas em um elemento combustível do reator de pesquisas IEA-R1 durante evento de perda lenta de vazão com RELAP", **Peer Review**, vol. 5, nº 18, p. 245–267, ago. 2023.

[8] WANG, M.; WANG, Y.; TIAN W.; QIU, S.; SU, G. H. "Recent progress of CFD applications in PWR thermal hydraulics study and future directions", **Annals of Nuclear Energy**, vol. 150, p. 107836, jan. 2021, https://doi.org/10.1016/j.anucene.2020.107836.

[9] SCURO, N. L. "Simulação numérica de um acidente tipo perda lenta de vazão em um reator nuclear de pesquisa", Mestrado em Tecnologia Nuclear - Reatores, **Universidade de São Paulo**, São Paulo, 2019. doi: https://doi.org/10.11606/D.85.2019.tde-25102019-154117.

[10] ANSYS CFX® 25.0, "User Manual", **ANSYS Europe Ltd.**, 2025.

[11] SALAMA, A. “CFD investigation of flow inversion in typical MTR research reactor undergoing thermal–hydraulic transients”, **Annals of Nuclear Energy**, vol. 38, nº 7, p. 1578–1592, jul. 2011, https://doi.org 10.1016/j.anucene.2011.03.005.

[12] KHATER, H.; EL-MORSHEDY, S. E.-D.; ABDELMAKSOUD, A. “Simulation of unprotected LOFA in MTR reactors using a mix CFD and one-d computation tool”, **Annals of Nuclear Energy**, vol. 83, p. 376–385, set. 2015, https://doi.org/10.1016/j.anucene.2015.03.041.

[13] HEDAYAT, A.; DAVARI, A. “Feasibility study to increase the reactor power at natural convection mode in Tehran Research Reactor (TRR) through a hybrid thermal-hydraulic simulation and analysis using the RELAP5 code and Computational Fluid Dynamic (CFD) modeling by ANSYS-FLUENT”, **Progress in Nuclear Energy**, vol. 150, p. 104285, ago. 2022, https://doi.org/10.1016/j.pnucene.2022.104285.

[14] AUMILLER, D.L.; TOMLINSON, E.T.; BAUER, R.C. A coupled RELAP5-3D/CFD methodology with a proofof-principle calculation, **Nuclear Engineering and Design**, Volume 205, Issues 1–2, 2001, Pages 83-90, ISSN 0029-5493, https://doi.org/10.1016/S0029-5493(00)00370-8.

[15] YE, L.; YU, H.; WANG, M.; WANG, Q.; TIAN, W.; QIU, S.; SU, G.H. CFD/RELAP5 coupling analysis of the ISP No. 43 boron dilution experiment, **Nuclear Engineering and Technology**, Volume 54, Issue 1, 2022, Pages 97-109, ISSN 1738-5733, https://doi.org/10.1016/j.net.2021.07.047.

[16] HA, T.; GARLAND W. J. “Hydraulic study of turbulent flow in MTR-type nuclear fuel assembly”, **Nuclear Engineering Design**, vol. 236, nº 9, p. 975–984, 2006, doi: https://doi.org/10.1016/j.nucengdes.2005.10.004.

[17] NUCLEAR AND ENERGY RESEARCH INSTITUTE (IPEN), “Relatório de Análise de Segurança IEA-R1: Capítulo 16 - Análise de Segurança”, 2017.

[18] CELIK, I. B.; GHIA, U.; ROACHE, P. J., FREITAS, C. J.; COLEMAN, H.; RAAD. “Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications”, **Journal of Fluids Engineering**, vol. 130, nº 7, p. 078001, 2008, https://doi.org/10.1115/1.2960953.

[19] INTERNATIONAL ATOMIC ENERGY AGENCY (IAEA), “Research Reactor Core Conversion Guidebook - Voume 2: Analysis”, Vienna 1992.

[20] NUCLEAR AND ENERGY RESEARCH INSTITUTE (IPEN), "**Documento de Qualificação - Elemento Combustível Padrão IEA-208**", Centro Do Combustível Nuclear - CCN, jan. 2010.

[21] WAGNER, W.; PRUß, A. "The IAPWS Formulation 1995 for the Thermodynamic Properties of Ordinary Water Substance for General and Scientific Use", **Journal of Physical and Chemical Reference Data**, vol. 31, nº 2, p. 387–535, jun. 2002, https://doi.org/10.1063/1.1461829.

LICENSE