SAUDI INTERNATIONAL CONFERENCE ON NUCLEAR POWER ENGINEERING

Proceedings of SCOPE
13-15 Nov. 2023 – KFUPM
Paper 23057

Numerical activities in support of the development of GEN IV LMFBRs at the University of Pisa: a review of recent works

Pietro Stefanini[1], Andrea Pucciarelli[1], Francesco Galleni[1], Nicola Forgione[1]
1 Università di Pisa, Dipartimento di Ingegneria Civile e Industriale, Largo Lucio Lazzarino 2, 56122 Pisa, Italy
Email(s): andrea.pucciarelli@unipi.it

Abstract – *Liquid Metal Fast Breeder Reactors (LMFBRs) represent one of the most promising proposals for the upcoming GEN IV of nuclear power plants. They indeed allow for both breeding processes and increased plant efficiencies: nevertheless, several challenges still need to be overcome. During the last years the European Union launched several projects in support of the development of such a technology: the University of Pisa joined the common effort providing numerical analyses addressing the thermal-hydraulics aspects of LFMBRs.*

In particular, system thermal-hydraulics codes and CFD approaches were considered for the analysis of both normal operating conditions and accidental scenarios. Buoyancy induced phenomena were particularly addressed aiming at understanding the capabilities of passive cooling systems. Both forced and natural circulation conditions were investigated: the results of the calculations were validated and compared against available experimental results showing in general good predicting capabilities.

The present paper reports on the recent numerical activities performed at the University of Pisa in support of GEN IV LMFBRs. The addressed experimental facilities and experimental data are presented discussing the limits and capabilities of the adopted modelling techniques being STH, CFD and coupled STH/CFD applications. The obtained results are considered as a basis for the suggestion of best practice guidelines for the simulation of some of the NPP primary system components paying particular attention to the required computational resources and expected/required refinement of the adopted model.

Keywords: Liquid Metals, CFD, STH, Coupled STH/CFD

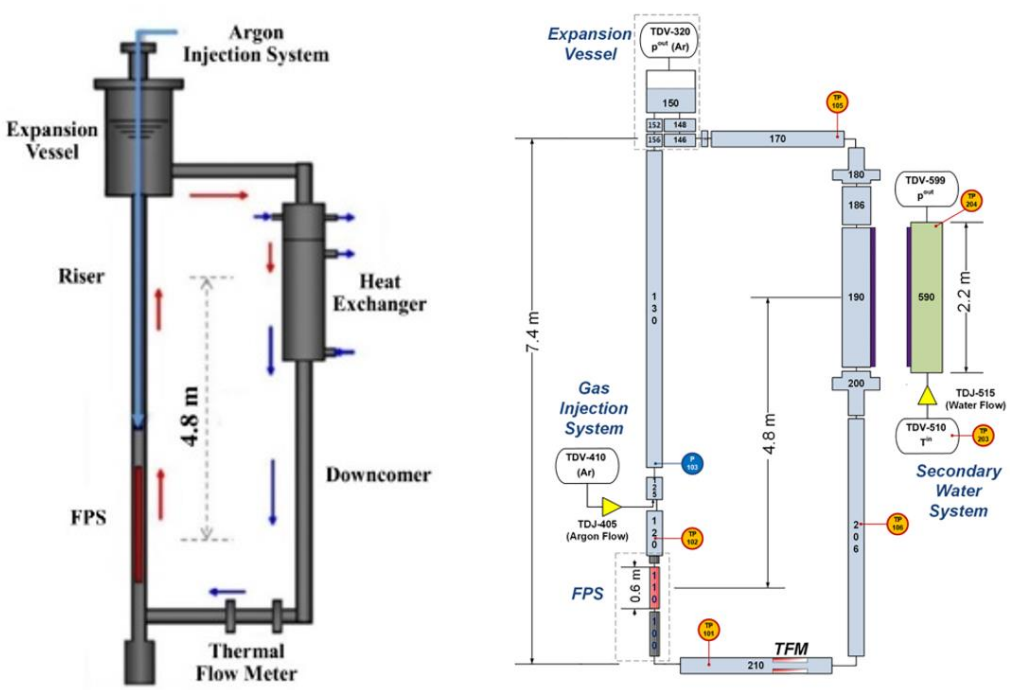
I. Introduction

In the global context, the production of energy from low-emission sources is becoming a crucial goal for the energy production industry especially due to the growing energy demand worldwide. Consequently, there has been a notable increase in interest in fission nuclear power plants in recent years since its emission-to-power ratio is one the most favourable. In fact, the high-power output and comparatively greenhouse gases lower emissions make fission nuclear power plants an attractive choice in the pursuit towards clean and sustainable energy sources. This paved the way for the interest in the development of new technologies, specifically the GEN IV's reactors, which introduce several advantages in terms of efficiency, waste management and safety with respect to the previous NPPs generations. To promote and advance this promising technology, the European Union started several projects over the past decades. Some examples include MYRTE[1], SESAME[2], and PATRICIA[3], which aim to coordinate collaborative efforts towards the development of LMFBRs. As part of this research, the University of Pisa has actively participated by contributing to the common effort with the development of numerical schemes and models addressing the thermal-hydraulics aspects of LMFBRs. The numerical approach adopted at the University of Pisa involved three main analysis approaches: System Thermal-Hydraulic (STH), Computational Fluid Dynamics (CFD) and coupled STH/CFD calculations. These approaches were employed to investigate various operating conditions, including normal and accidental scenarios, with a specific focus on understanding the buoyancy-driven passive cooling mechanisms. Additionally, transient scenarios were considered to study the transition from forced to natural circulation. The adopted and developed numerical models demonstrated promising results. Validation was mainly performed by comparing numerical results with data obtained from the experimental facilities, and, in general, the results showed good agreement and thus providing room for future applications for safety analyses. The present paper resumes the numerical activities performed at the University of Pisa in support of GEN IV LMFBRs. Each section addresses a specific experimental facility showing the adopted modelling tools and the obtained results. The lessons drawn are considered as a basis for the suggestion of best practice guidelines for the simulations.

II.NACIE-UP

The NACIE-UP facility [4] is a rectangular loop set at ENEA Brasimone RC that has been specifically designed to investigate thermal-hydraulic phenomena involving liquid metals, both in forced and natural circulation conditions. The primary working fluid within the loop is LBE. Figure 1A reports a sketch of the addressed facility, highlighting some of the most important components. In particular, it must be mentioned that the circulation can be promoted by an Argon injection system located at the top of the Fuel Pin Simulator (FPS) region: consequently, no mechanical pump is needed to achieve forced circulation conditions. Among the various components, the FPS holds particular significance. This component, built in similarity with the expected MYRRHA fuel elements, supplies power to the system and, additionally, the presence of wire-wrapped spacers induces significant cross-sectional flows, adding complexity to the system and making 1-D numerical approaches less effective in addressing this specific component.

On the other hand, though the FPS presents a complex 3D environment, the whole facility is mainly mono-dimensional and represents a valuable environment for the validation of STH codes. This is e.g. the case of the numerical analyses [4] performed in the frame of a benchmark launched during the SESAME project. This benchmark study focused on three specific tests: Gas-lift transition, Power transition, and a PLOFA (Protected Loss Of Flow Accident) scenario. The aim of the cited work was to reproduce these tests using a standalone STH calculation approach, employing RELAP5/Mod3.3 code. The validation of the models was performed by comparing key parameters such as the inlet and outlet temperatures of the FPS and of the Heat eXchanger



(A) (B)

Figure 1: (A) Facility sketch. (B) Facility nodalization [5]

(HX), and the mass flow rate within the loop. The entire domain of the facility was replicated in the STH domain, adopting the nodalization reported in Figure 1B. The results presented a sufficient agreement with the experimental results, underlining that the STH approaches could be used to represent a variety of conditions, both operational and accidental, for facilities like NACIE-UP. The investigation of local phenomena occurring inside the FPS region needs instead a more detailed simulation domain. In the context of SESAME project, the NACIE-UP facility was utilized to investigate the occurrence of local peak temperatures resulting from flow blockages within the rod bundle. During the experimental campaign, different levels of occlusion were studied. The University of Pisa opted for a CFD RANS/URANS approach. In this study [6] the FPS domain was accurately represented within the CFD domain to study localized phenomena that may occur during flow blockage accidents. In particular, two experimental cases were considered trying to find the most suitable model applicable to this kind of scenario. In the CFD domain, a highly refined model was constructed including also the solid region. A mesh independence analysis was conducted to ensure the reliability of the computational model. Notably, interesting results were obtained by comparing CFD simulations with experimental data. As reported in Figure 2, two different calculation procedures were employed in the analysis: RANS (Reynolds-Averaged Navier-Stokes) and URANS (Unsteady Reynolds-Averaged Navier-Stokes). Both approaches were able to capture the peak temperature in the recirculation region immediately after the flow-blockage, but tended to overestimate it. Nevertheless, the URANS calculation exhibited a better trend and closer agreement with experimental data, suggesting that a more refined model like LES could improve the results.

To develop a coupling procedure between STH/CFD codes, the FPS region was extensively studied in further works. In the work [7], the FPS regions were simulated in the CFD domain using two different software packages: ANSYS Fluent and STAR-CCM+. The objective of this analysis was to assess the capabilities and uncertainties associated with the available CFD codes and techniques in the field of liquid metal thermal hydraulics. In the aforementioned study, the numerical results obtained from the CFD simulations were compared against experimental data. Additionally, the Cheng and Todreas correlation, which demonstrated interesting capabilities in previous research works, was also considered for the prediction of pressure drops across the component. The findings of the study indicated that both ANSYS Fluent and STAR-CCM+ provided good estimations of the pressure drops within the FPS region. Particularly, when adopting the SST k-ω turbulence model the simulations showed better agreement with the selected correlation. This matching suggests promising capabilities in view of applications involving Fluent/REALAP5 coupled calculations: a suitable estimation of the pressure drops in the FPS

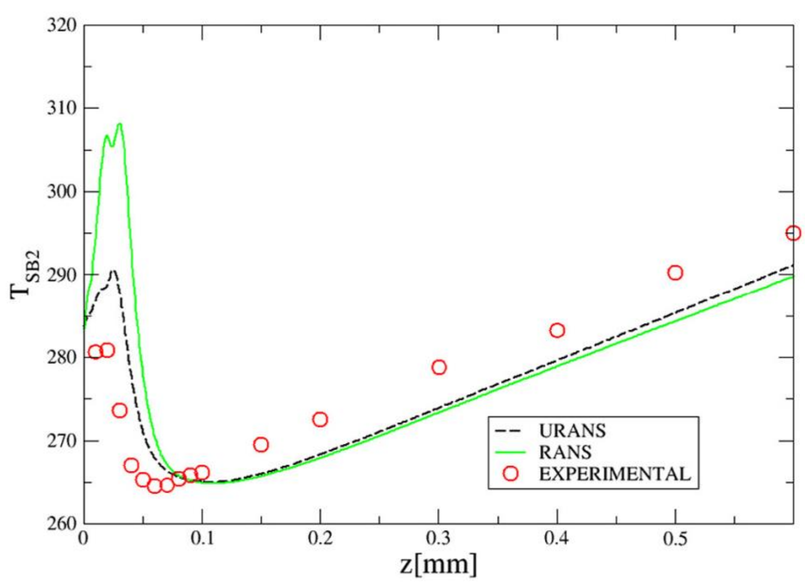


Figure 2: Comparison between experimental data and numerical result for local temperature in the Pin Bundle[7]

region could indeed really improve the quality of the obtained results.

The coupling approach was adopted for some selected PLOFA scenarios [8]. For this application, the overall system was simulated using the STH code RELAP5, while the FPS region was simulated using the CFD code ANSYS Fluent. Figure 3 presents the spatial coupling scheme used for this specific application. It outlines the spatial arrangement and integration of the STH and CFD domains within the simulation setup. The coupling scheme enabled the exchange of information and interaction between the two codes, allowing for a comprehensive analysis of the PLOFA scenarios.

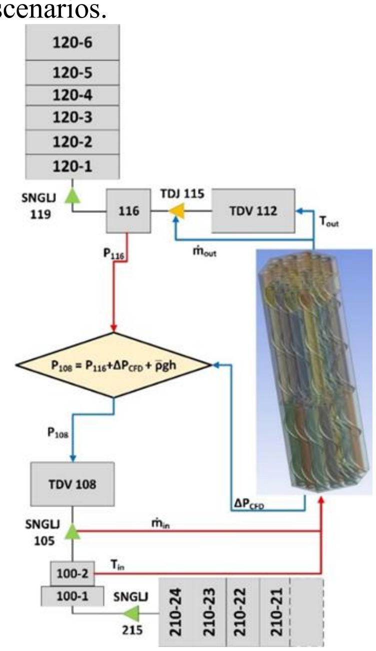


Figure 3: spatial coupling scheme for the Pin Bundle[8]

By combining the capabilities of RELAP5 and ANSYS Fluent, this coupling procedure provided a more accurate representation of the thermal-hydraulic behaviour during the PLOFA events. The STH code captured the system-level response while the CFD code offered detailed insights into the localized phenomena within the FPS region. This integration of different codes and approaches enhanced the overall understanding and prediction capabilities for the analysis of complex accidental scenarios, being able to both represent global trends and catch local phenomena. Indeed, Figure 4, as presented in the study, shows the wall temperature predictions provided by the coupled STH/CFD calculations for the analysed PLOFA scenario. The results exhibit a good level of agreement with the experimental data, highlighting the effectiveness of the coupling approach. Unlike a standalone STH application, which can only predict cross-sectional averaged quantities, the coupled approach offers insights into subchannel bulk temperatures and the local behaviour within components. This enables a more comprehensive understanding of the system's thermal-hydraulic response during PLOFA events. This interaction between the codes enhances the overall predictive capabilities, merging the ability of STH code in the prediction of the global behaviour, with the capability of CFD in representing complex local phenomena. This provides valuable information for safety assessments and design optimizations in complex thermal-hydraulic systems.

III. CIRCE-HERO

The CIRCE-HERO facility is an integral test section set at the ENEA Brasimone Research Centre specifically designed for the thermal hydraulic phenomena involving Led Bismuth Eutectic (LBE) [9]. The facility can be described focusing on two main

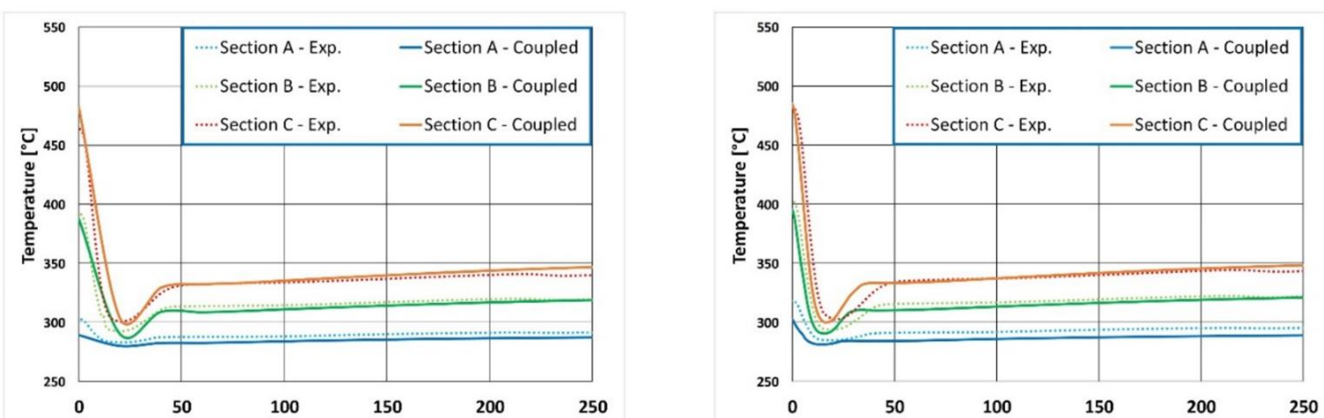


Figure 4: Wall temperature predictions provided by coupling procedure [8]

parts: the CIRCE-pool and the HERO-loop. The CIRCE-pool is a stainless-steel cylinder approximately 8500 mm tall having a radius of about 1170mm. This cylinder houses the HERO-loop, which provides room for the circulation of LBE within the system. Figure 5 reports a scheme of the facility and the LBE main flow path.

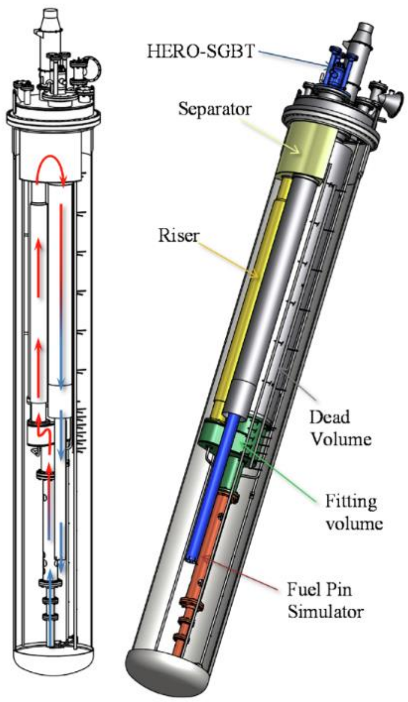


Figure 5: CIRCE-HERO facility scheme[9]

During the last years the University of Pisa was involved in some of the experimental campaigns focused on the CIRCE-HERO facility. These experimental campaigns served as valuable platforms for testing and validating the developed numerical models and approaches.

In the work by K. Zwijesn et al.[10], a comparison was made between two in-house coupling schemes: one from NRG and the other from the University of Pisa. Focusing on the University of Pisa's approach, coupled RELAP5/Mod3.3 and ANSYS Fluent calculations were performed. The STH approach was used to simulate the HERO loop and the secondary side, while the 3D approach, i.e. the CFD model, was used to describe the CIRCE-pool environment. The coupling was applied to both steady-state and accidental scenarios, with a focus on 3D fluid flow, thermal stratification, and temperature changes during the addressed transients. The results show a good match between the flow and temperature evolution and, the experimental data. The University of Pisa also conducted CFD-standalone calculations to better understand local phenomena within the pin bundle of the FPS [11]. This approach provided detailed insights into fluid flow patterns and temperature distribution at a localized scale, complementing the overall investigation of the coupled calculations. In similarity with what was done for the NACIE-UP facility, RANS and URANS calculations were adopted to achieve a more detailed analysis of some key components. Particular attention was paid to the investigation of the heat exchange between FPS and the pool during the considered transient cases. From the comparison with experimental data, it turns out that for the steady state case, heat transfer towards the pool doesn't play a relevant role in the overall FPS mechanisms, while it becomes relevant during the addressed transient cases. In fact, during the transient, the boundary conditions to be imposed on the external walls of the FPS become a very important aspect and key parameter of the calculation. In fact, they allow the simulation the thermal inertia of the pool, which affects the temperature distribution inside the FPS while the heating power is turned off. The work demonstrated that the contribution provided by the heat exchange with the pool was fundamental to reproduce the experimental FPS outlet temperature trend during the transient. Looking at the temperature distribution along the FPS, good agreement was found in the prediction of local effects due to the presence of spacer grids, as it can be seen from Figure 6.

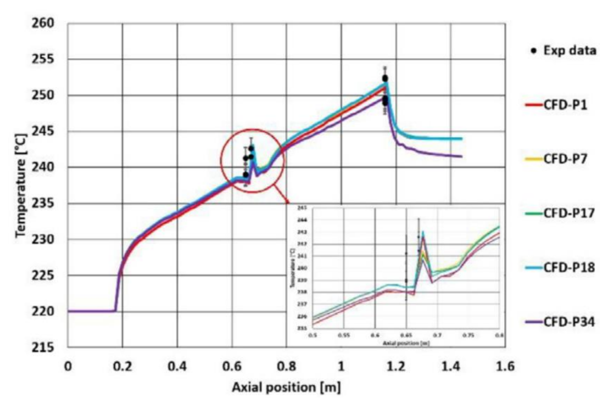


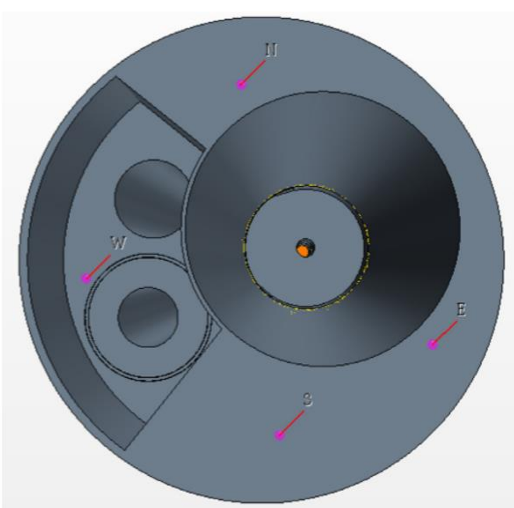
Figure 6: Prediction of local effects along the FPS[11]

In another study conducted at the University of Pisa [12], researchers focused instead on the examination of the thermal-hydraulics of the CIRCE-pool environment. To achieve this, two different CFD codes were used: the ANSYS Fluent and the STAR-CCM+ commercial codes. In both cases, steady-state RANS calculations were performed. The simulated geometry consisted of the fluid region of the pool, while the loop region was considered as an empty space, setting suitable boundary conditions derived from experimental data. The numerical results were then compared with experimental data. Particular attention was paid to the analysis of the thermal stratification along the vertical axis of the pool. Figure 7 displays the outcomes obtained in this research, demonstrating a good agreement between the simulation and experimental data. This successful match paved the way for future coupled calculations of the entire facility.

In fact, the University of Pisa later developed a new coupling procedure to better comprehend the behaviour of the key components of the CIRCE-HERO facility, specifically focusing on the HX-HERO bayonetted tube steam generator [13]. In this study, the primary side was replicated within the computational fluid dynamics (CFD) domain, while the two-phase mixture of the secondary side was reproduced within the system thermal-hydraulic (STH) domain. The coupling process involved the use of two different codes, namely ANSYS Fluent and RELAP5/Mod3.3. The model was designed with a specific approach: the STH domain provided the estimation of the bulk temperature and heat transfer coefficient (HTC) of the ascending water, which was then supplied to the CFD domain. On the other hand, the CFD provided the wall temperature at the LBE side surface of the pipes back to the STH domain. The numerical results obtained from this coupling procedure were then compared to several operational conditions of the same component. The predictions demonstrated a strong qualitative agreement with the experimental data, indicating the effectiveness and general applicability of this methodology.

In addition, the environment of CIRCE-HERO was deeply investigated under many points of view before proceeding with a coupling scheme. A standalone STH approach was presented in the work of [14] where PLOFA scenarios were simulated. After reproducing the facility through an accurate nodalization, boundary conditions were applied according to the experiments. The overall numerical results underlined a good agreement with the experimental data. Nevertheless, the pool environment nodalization developed for the RELAP5 environment lacked the detailed description provided by CFD calculations. The very same PLOFA was thus analysed also in the work [15] applying a coupling scheme. Figure 8 shows a comparison between the results nodalizations adopted by the two numerical approaches.

The coupled scheme allowed achieving a good matching with experiments for loop's temperature and mass flowrate. Also, the temperature distribution along the vertical axis of the pool was well matched. In addition, also interesting behaviours occurring inside the pool, e.g. the recirculation phenomena occurring at the bottom, could be highlighted thanks to the CFD contribution. All this work presented a valuable ground on which future works could base its roots.



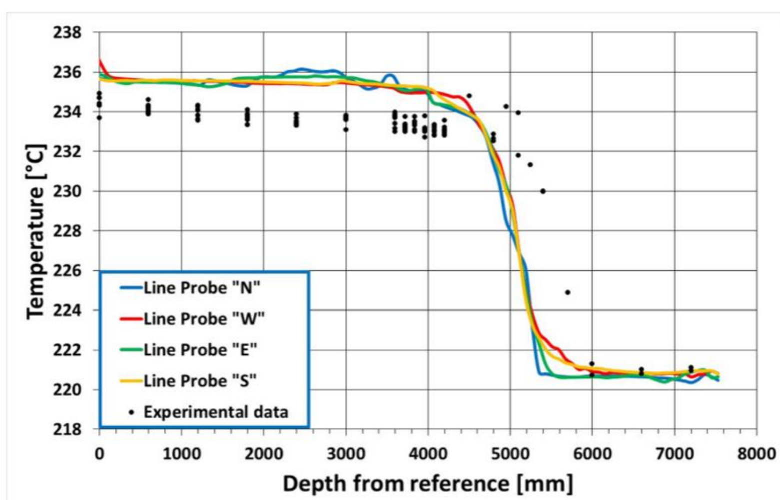


Figure 7: Temperature distribution along the vertical axes inside the CIRCE pool [12]

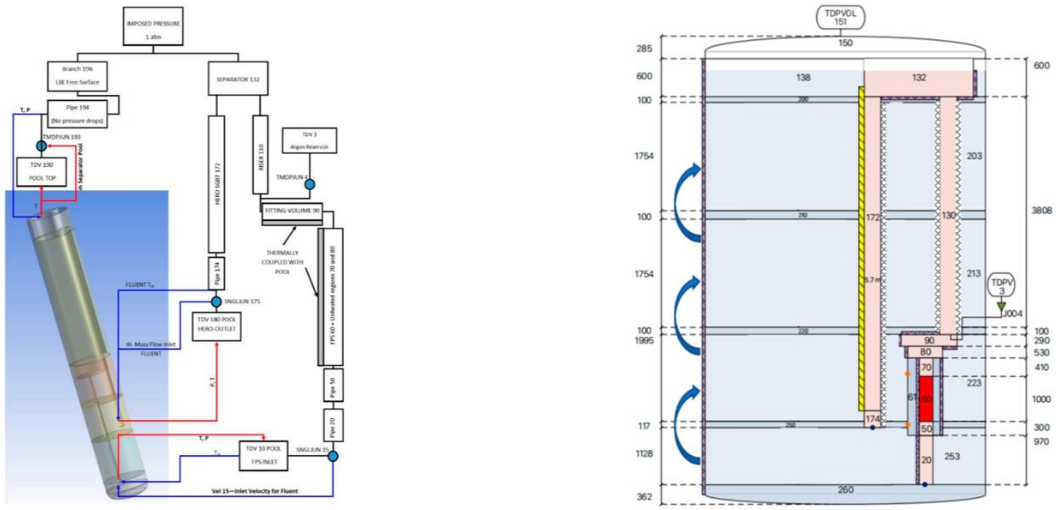


Figure 8: Comparison between numerical coupling scheme and, STH standalone computing domain [14][15]

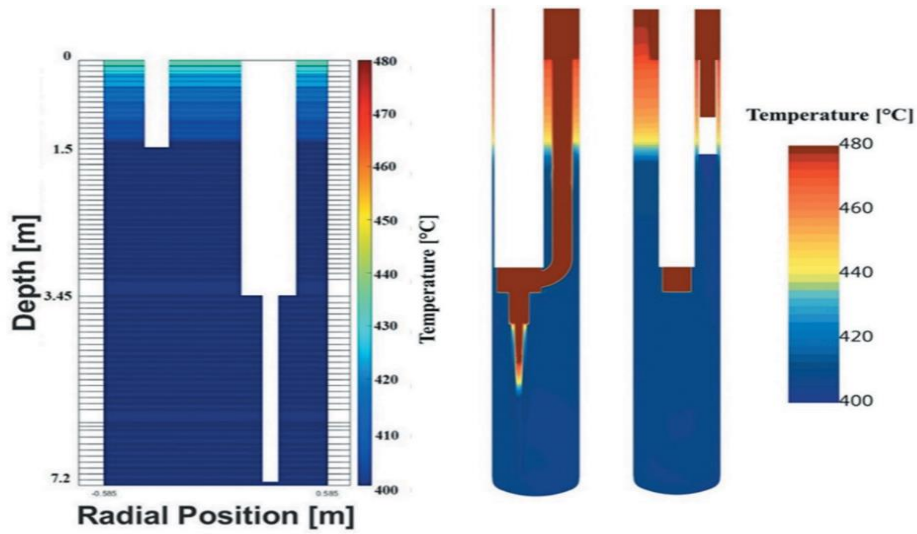
IV. CIRCE-THETIS

Concerning current and future works to be performed at the University of Pisa, the analysis of the CIRCE-THETIS facility is among the foreseen activities for the current EU-PATRICIA project. With respect to the previous CIRCE-HERO configuration, the main difference stands in the HX length and geometry. The shorter HX length enhances the mixing phenomena reducing thermal stratification and enhancing the heat exchange between the pool and loop. Figure 9 shows the comparison between the predictions provided by STH and CFD approaches for the axial thermal stratifications occurring inside the pool. As it can be seen from this figure, both the approaches predict a higher positioning of the thermal stratification with respect to the CIRCE-HERO configuration. In order to better understand the thermal behaviour inside the pool, some preliminary works were performed at the University of Pisa [16] and [17] providing pre-test calculations trying to understand the best way to simulate this facility. Pre-test analyses were performed adopting the CFD codes ANSYS Fluent and STAR-CCM+ and the STH codes RELAP5/Mod3.3 and RELAP5-3D. STH calculations highlighted some limits in predicting the thermal-hydraulic behaviour of the pool environment owing to the involved complicated 3D mixing phenomena that cannot reliably be predicted by this approach. The CFD approach should thus instead be adopted for the pool region. On the other hand, the two-phase flow conditions in the secondary side need an STH approach to be reproduced. As a consequence, in future works coupled STH/CFD schemes will be applied to this new configuration to perform pre-test and post-test analyses.

V. Conclusions

The current work summarized the outcomes of the recent STH, CFD and STH/CFD simulations performed at the University of Pisa. The advantages and drawbacks of the addressed approaches are discussed as well.

The STH approach proved to be a valuable tool for the analyses of large facilities and long transients. Unfortunately, it highlighted several limits when large 3-D environments had to be addressed. Its superiority in dealing with two phase-flow conditions must be acknowledged. On the other hand, CFD showed interesting capabilities in dealing with 3-D environments and allowed the analysis of local phenomena that could not be predicted by adopting the STH approach. Among its limits, there are the larger computational cost and its unsuitability for the analysis of two-phase flow conditions. Coupled STH/CFD calculations thus represent a valuable analysis tool allowing overcoming the limits of both the approaches and trying to keep only their advantages. The obtained results showed promising capabilities that could be further exploited for the safety analyses of GEN IV NPPs. All the cited approaches will be adopted in the frame of future calculations, trying to prepare general guidelines for their application in the nuclear field and thus pave the way for the development of GEN IV LMFBRs.



(a) (b)

Figure 9: Comparison between STH and CFD Result of CIRCE-THETIS [17]

VI. References

[1] MYRTE, http://myrte.sckcen.be
[2] SESAME, http://sesame-h2020.eu
[3] PATRICIA, https://patricia-h2020.eu
[4] Di Piazza, I., Angelucci, M., Marinari, R., Tarantino, M., Forgione, N., 2016. Heat transfer on HLM cooled wire-spaced fuel pin bundle simulator in the NACIE-UP facility. Nucl.
[5] Eng. Des. 300, 256–267. N. Forgione, D. Martelli, G. Barone, P. Lorusso, T. Hollands, A. Papukchiev, M. Polidori, A. Cervone, I. Di Piazza "Post-test simulations for the NACIE-UP benchmark by STH codes", Nuclear Engineering and Design, 353 (2019) 110279.
[6] R. Marinari, I. Di Piazza, M. Tarantino, N. Forgione, "Blockage fuel pin simulator experiments and simulation", Nuclear Engineering and Design 353 (2019) 110215.
[7] A. Pucciarelli, G. Barone, N. Forgione, F. Galleni, D. Martelli, "NACIE-UP post-test simulations by CFD codes", Nuclear Engineering and Design, 356 (2020) 110392
[8] A. Pucciarelli, F. Galleni, M. Moscardini, D. Martelli, N. Forgione, "STH/CFD coupled calculations of postulated transients from mixed to natural circulation conditions in the NACIE-UP facility, Nuclear Engineering and Design, 370 (2020) 110913
[9] Pesetti, A., et al., 2018, "ENEA CIRCE-HERO test facility: geometry and instrumentation description", ENEA report CI-I-R-343, June 2018.
[10] K. Zwijsen, D. Martelli, P.A. Breijder, N. Forgione, F. Roelefs Multi-scale modelling of the CIRCE-HERO facility. Nucl. Eng. Desing, 355 (2019) (2019), p. 110344
[11] F. Buzzi, A. Pucciarelli, F. Galleni, M. Tarantino, N. Forgione, "Analysis of the temperature distribution in the pin bundle of CIRCE facility", Annals of Nuclear Energy, 147 (2020) 107717
[12] F. Buzzi, A. Pucciarelli, F. Galleni, M. Tarantino, N. Forgione, "Analysis of thermal stratification in the CIRCE-HERO facility, 141 (2020) 107320.
[13] F. Galleni, G. Barone, D. Martelli, A. Pucciarelli, P. Lorusso, M. Tarantino, N. Forgione, "Simulation of operational conditions of HX-HERO in the CIRCE facility with CFD/STH coupled codes", Nuclear Engineering and Design, 361 (2020) 110552.
[14] M. Moscardini, F. Galleni, A. Pucciarelli, D. Martelli, N. Forgione, "Numerical Analysis of the CIRCE-HERO PLOFA Scenarios, Applied Sciences 2020, 10 75358, DOI: 10.3390/app10207358
[15] A. Pucciarelli, F. Galleni, M. Moscardini, D. Martelli, N. Forgione, "STH/CFD Coupled simulation of the Proctected Loss of Flow Accident in the CIRCE-HERO Facility" Applied Sciences 2020, 10, 7032; doi:10.3390/app10207032
[16] P. Stefanini, A. Pucciarelli, N. Forgione, I. Di Piazza, D. Martelli, "Numerical analyses of the CIRCE-THETIS facility by means of STH and CFD codes" Nuclear Engineering and Design 409 (2023) 112349.
[17] P. Stefanini, F. Galleni, I. Di Piazza, A. Pucciarelli, "Liquid-Metal Thermal-Hydraulic Numerical Analyses in Support of the Upcoming CIRCE-THETIS Exerimental Campaign, Nuclear Technology, DOI: 10.1080/00295450.2023.2189892