# THE ROLE OF CFD SIMULATIONS IN CRS4 SUPPORT TO LIQUID METAL FAST REACTORS DESIGN

*MANUELA PROFIR and VINCENT MOREAU*

CRS4, Sardinia, Italy
*e-mail: manuela@crs4.it*

*Abstract*

Thanks to the opportunity of being partner in several EURATOM projects, CRS4 developed many complex CFD applications in the innovative nuclear systems field. We illustrate the CFD model of the primary coolant loop of MYRRHA, built in the SEARCH and MYRTE projects. Well aware that the modelling needs to be validated, we compared the numerical results with relevant data, obtained with experimental dedicated campaigns: the Control Rod insertion in the MAXSIMA project and the simulations in the SESAME-Stand experimental facility, investigating the lead solidification. In both cases, the comparison has brought decisive modelling improvement and better understanding of the experimental results. The last application regards the CFD modelling of the ALFRED primary coolant loop, strongly parameterized so as to investigate several possible design variants. The control demonstrated on the CFD modelling thanks to the EURATOM projects is allowing CRS4 to propose itself as a complementary partner in the network of specialists supporting the design of innovative GEN-IV nuclear systems.

***Keywords:*** *CFD simulation, Solidification, MYRRHA, ALFRED, Control rod*

## Introduction

In the last 20 years, the Sardinian Research Center CRS4 (Centro di Ricerca, Sviluppo e Studi Superiori in Sardegna) has developed a consolidated experience in performing Computational Fluid-Dynamics (CFD) simulations in the field of innovative nuclear systems. Currently, CFD is showing its ability to capture a series of features that are difficult to observe or reproduce with system codes. It has thus the pretention to become another and a complementary tool in the designer hands.

Thanks to the opportunity of being partner in several EURATOM projects (CDT, SEARCH, MAXSIMA, SESAME, MYRTE), the complexity of the applications developed by CRS4 has progressively increased during these years and some of the most representative ones are presented in the following.

We start by illustrating the CFD model of the entire primary coolant loop of MYRRHA (Multi-purpose hYbrid Research Reactor for High-tech Applications), the future Belgian multi-purpose Lead-Bismuth Eutectic (LBE) cooled nuclear facility, built in the framework of the FP7 SEARCH project and continued in the H2020 MYRTE and SESAME projects. The focus is put on the strong multi-physics character of the CFD modelling able to simulate scenarios with variation of the free surface levels as well as operational incidents or accidents like fuel dispersion or diaphragm rupture.

To validate the modelling, the numerical results have been compared with relevant data, obtained during experimental campaigns performed in dedicated facilities. In this sense, it can be mentioned the work on the Control Rod insertion performed in the COMPLOT (COMPonent LOop Testing) facility in close collaboration with SCK•CEN, the Belgian Nuclear Research Center, during the MAXSIMA project.

Similarly, pre-test and post-test simulations have been performed according to the experiments done by the Research Centre Rez (CVR) in the Sesame-Stand facility, built to investigate the solidification and the re-melting of the lead in a buoyant vessel. In both cases, the comparison has brought decisive improvement of the modelling and also a better understanding of the experimental results.

The last application takes profit of the recent past experience and is currently on-going. It regards the modelling of the ALFRED primary pool, re-investigated in the final part of the SESAME project, and whose upgraded version has been released during the SESAME International Workshop in March 2019. Being in a preliminary phase, the CFD model, mainly the building of the ALFRED geometry, has been strongly parameterized so as to investigate several possible variants of the design and of the working operation.

## CFD Model of MYRRHA Primary Coolant Loop in SEARCH and MYRTE

Following the work performed in several European Projects from the 5$^{th}$ Framework Program (FP5) on, the FP7 SEARCH and the H2020 MYRTE/SESAME projects supported the development of MYRRHA.

Within these projects, the CRS4 objective was to qualify the CFD as a valid instrument for the design, verification and analysis of the complex flow patterns of the MYRRHA primary system in various operating conditions: normal, incidental and accidental. The CFD model of the MYRRHA primary system made within the STAR-CCM+ simulation code [1] is illustrated hereafter.

The CFD tool was used at first during the FP5 EUROTRANS project, restricted to the study of some parts of nuclear reactor designs, frequently taking advantage of the symmetries [2]. However, this approach limited the range of applications, forbidding the investigation of asymmetric events and transients.

A first entire CFD model of MYRRHA primary coolant loop has been developed during the SEARCH FP7 program with the main purpose to investigate possible fuel dispersion and re-aggregation [3, 4]. The model was based on the VOF (Volume of Fluid) paradigm, with the cover gas and the main structures included. It has proved to be fully compatible with the use of the passive scalars that allowed determining some relevant characteristic times. An evaluation of the dispersion of Lagrangian particles representative of fuel particles with different size and density under nominal operation for up to 1200s was possible. In particular, there is no re-aggregation of the heavier particles at the vessel bottom, as they are re-entrained by the pump flow. The larger heavy particles soon finish at the bottom of the upper plenum. The model was also upgraded to allow simulation of an Unprotected Loss of Flow (ULOF) scenario with consequent release of fission gas and fuel particles (see Fig. 1).

To demonstrate the feasibility of such a scenario, it has been necessary to develop the framework of a preliminary coupling of the core power with the neutronic feedback due to the LBE change of temperature. Under the ULOF scenario, except the smaller ones, most of the heavier particles precipitate at the bottom of the hot plenum while the lighter particles remain trapped by the hot plenum free surface. The main issue addressed was the eventuality of a re-criticalisation under fuel dispersion events. A variety of behaviour has been observed, depending on the size and weight of the particles, with a large extension of the re-aggregation zones such that it is very unlikely that a high quantity of fuel can gather locally in one of these zones. In consequence, a re-criticalisation seems quite unlikely.

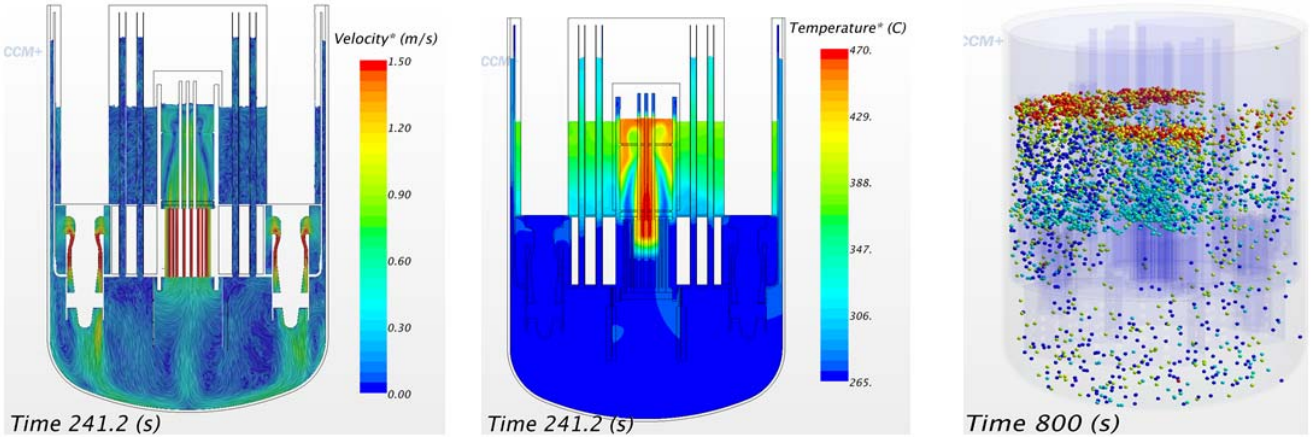



Fig. 1. Fission gas release and fuel particle release in ULOF incident

The work in MYRTE is based on the experience accumulated in SEARCH. Currently, CFD is the only tool able to capture the 3D flow effects in the plenum of a pool reactor. However, it still isn't feasible yet to simulate all the details of an entire reactor. For applications like safety studies and design, the exact geometrical representation of the core, heat exchangers, pumps and bypasses can be replaced by simplified models without loss of relevant information. Bypass flows have been implemented in the current model of MYRRHA [5] in order to connect the fluid volumes and to avoid stagnation regions, which could cause potential corrosion issues. With the bypass flows and the optimized ACS, the CFD model became quite suitable for nominal operation. Moreover, the VOF method allowed the simulation of scenarios with variation of the free surface levels, in particular, the simulation of a transient scenario involving the rupture of the diaphragm. A large area of both the upper and lower diaphragm plates is opened all of a sudden. This is typical of an event in which an initial fracture in the diaphragm plates propagates very fast so that a large breach is opened. Although the simultaneous appearance of two large breaches, in both lower and upper diaphragm plates, is quite unrealistic, the information obtained in this scenario could be used in the case of a vessel with only one solid interface between the hot and the cold plenum. The diaphragm rupture is not averted by the control system. The pumps keep working all the time at prescribed velocity, only slightly increasing the flow rate in front of a large decrease of the pressure drop. When a thermocouple above the fuel assemblies (FAs) registers a temperature increase by more than 20 K, after one second the shutdown procedure initiates, lasting another 1s, and then the core heat source follows the prescribed decay curve. Even if this scenario wasn't foreseen during the geometry building, geometrical decomposition of the computational domain in three macro regions allowed us to investigate it at the cost of minor changes. It turns out that this scenario is much less critical for the core integrity as initially feared because the pumps are still running and there is no change of flow direction in the lower plenum, the flow through the new pathway being parallel to the core flow (see Fig. 2).

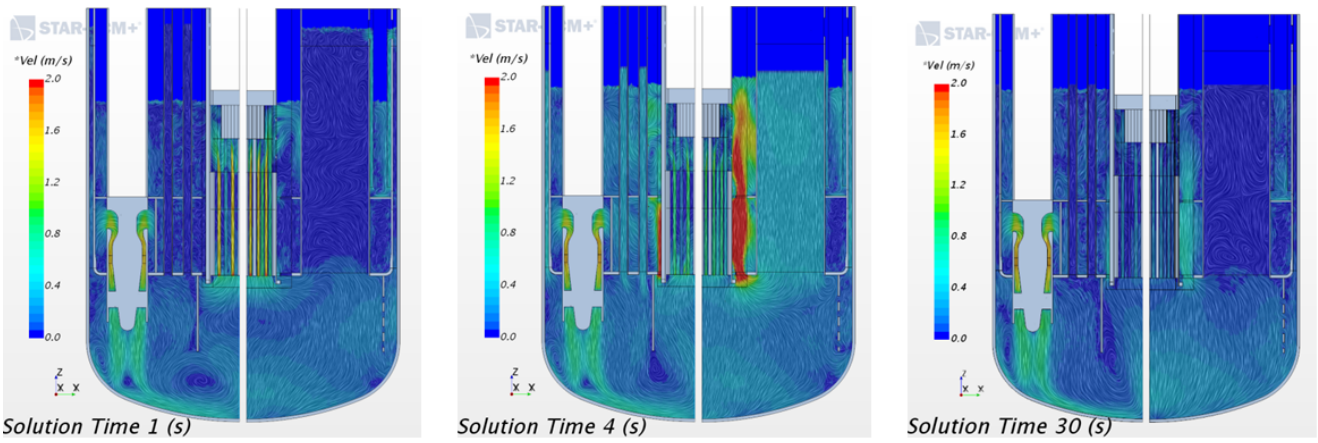



Fig. 2. Large loss of diaphragm integrity. Flow field at transient start, 3s later and at simulation finished

**Numerical and Experimental Campaign on CR insertion in LBE in MAXSIMA**

A combined numerical - experimental investigation campaign of the buoyancy driven MYRRHA control rod (CR) insertion during an emergency SCRAM, aimed to demonstrate its proof-of-principle, was performed in the framework of the MAXSIMA project [6].

A full-scale test section of the control rod was constructed at SCK•CEN to test the hydrodynamics in Lead Bismuth Eutectic over a wide range of operating conditions providing experimental data for the qualification of the CR system. A numerical CFD model of the CR test section was also setup in STAR-CCM+ for the pre-test and post-test analysis. Numerically, the CFD model requires application of the overset mesh method for the control rod effective insertion in an extremely constrained domain. The CR acceleration arises from Newton's law but also strongly affects the surrounding flow pressure, requiring a two-way coupled solution of the CR motion and the fluid dynamics. The CR dynamics is also controlled by distant boundary conditions which require the inclusion of several free surfaces; hence the VOF framework was employed.

The numerical model was progressively improved during the project also thanks to the constant feedback from the experimental on-going work [7]. When the numerical results were compared with the experimental results, the control rod dynamics showed similar behaviour, but still far from the experimentally measured curves (the numerical bundle insertion was about 15% faster than in the experiment). One likely explanation was the fact that the CFD model did not consider the possible bypass flow around the guide tube. While the CFD model considered only the guide tube, a labyrinth seal existed on the guide tube exterior. The presence of the bypass flow in the COMPLOT experiment meant that less mass flow passed through the experimental guide tube but it wasn't possible a quantification of it. The CR dynamics simulation was repeated with a reduced mass flow rate and the numerical results showed excellent agreement with the experimental results (Fig. 3). The numerical model was thus fully validated and, moreover, has provided additional information on the experimental bypass flow which, thanks to the numerical work, was evaluated to be about 20%.

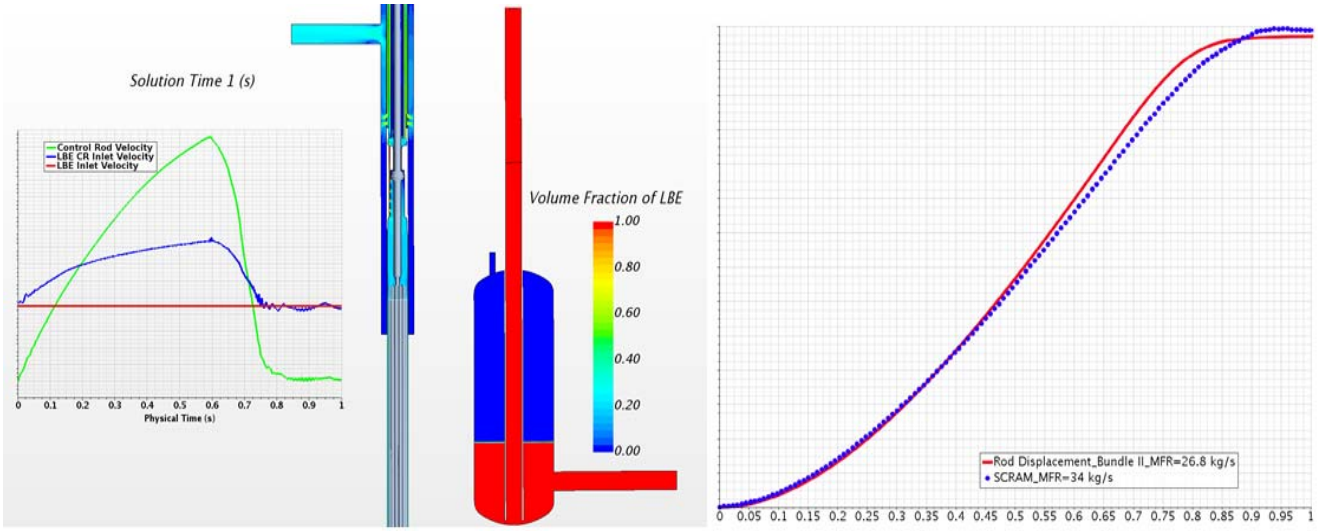



Fig. 3. CR insertion in COMPLOT test section: velocity (left); bottom tank buffer with free surface (centre) and numerical vs. experimental displacement curves (right)

**Numerical and Experimental Campaigns on Lead Solidification in SESAME**

The development and the validation of computational models able to simulate the liquid metal thermal-hydraulics and the relative freezing phenomena are essential for the development of liquid metal cooled Gen IV reactors and for their safety assessment. The coolant solidification is considered one of the basic thermal-hydraulics challenges [8] and a potential safety issue, since it may occur whenever an overcooling from the secondary system or the (passive) emergency cooling system is verified. The local solidification of the liquid metal can affect the natural convection pattern in the pool. In certain overcooling accidental scenarios, the liquid-metal coolant solidification can result in the reactor main flow path partial or complete blockage, consequently leading to core overheating. Solidification might also lead to mechanical stresses in components as a result of thermal contraction and expansion. Hence, it is important to know if and where solidification can occur during accidental transients and how the solidification front moves. In this sense, important activities focused on development and validation of computational models simulating the liquid lead natural convection and freezing have been performed within the SESAME project [9]. One of the CRS4 tasks in the ″Pool Thermal-hydraulics″ work package of this project was to perform advanced CFD simulations of the liquid lead circulation under freezing conditions, in collaboration with the Research Centre Rez (CVR). Therefore, we built a full 3D numerical model representative of the SESAME-Stand facility, in which CVR performed the solidification experiments campaigns. The model includes a pool type vessel equipped with four heating rods, with an internal obstacle supporting the natural convection and with overall distributed thermocouple probes. The external surface of the experimental vessel is cooled down by air with controllable mass flow rate which allows steady state and transient studies of the lead solidification under various conditions. The model was developed first as a pre-test model and in post-test phase, it was further analyzed and improved. Steady state and fast transient cases were reproduced according to the set of experimental tests performed at CVR. An investigation campaign was actuated in order to understand the numerical-experimental differences and improve the numerical model. Initially, the numerical results presented large differences with respect to the experimental ones. The first experimental results obtained in the SESAME-Stand facility showed that the heat losses were much more intense than expected from the simulation. The potential origin of these added losses had to be investigated. The most important contribution was given by the radiative thermal flux, not initially implemented in the model. Once done, the differences reduced drastically and a satisfying agreement was obtained. Progressively improving the thermal radiation modelling, the differences reduced significantly until an excellent agreement with the experimental results was reached, as illustrated in Figs. 4, 5 and 6, respectively. These pictures illustrate only a few simulated cases, while the detailed description of the model as well as the results obtained are included in the SESAME project deliverable D3.13 [10].

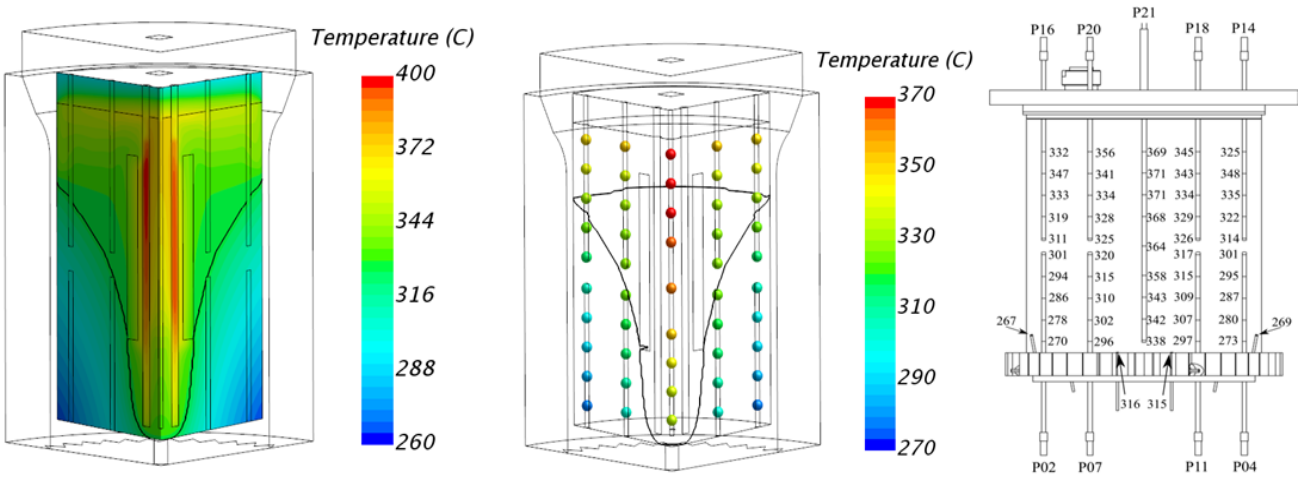



Fig. 4. Steady state temperature configuration with frozen fraction 58% in the numerical model (left and centre) against the experiment (right)

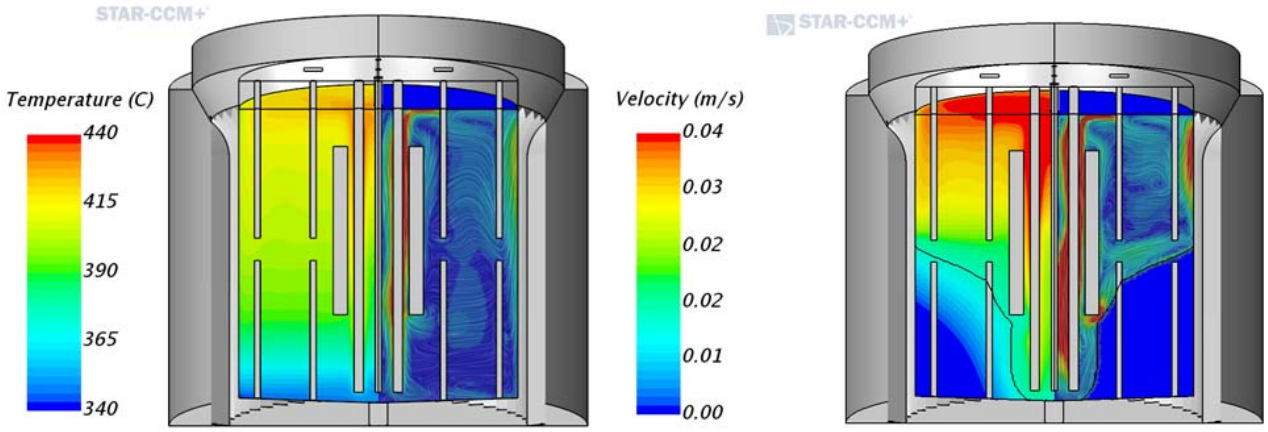



Fig. 5. Solidification transient simulation in SESAME-Stand. Initial steady state condition (left). Frozen fraction at 45% during re-melting phase after 2 hours (right)

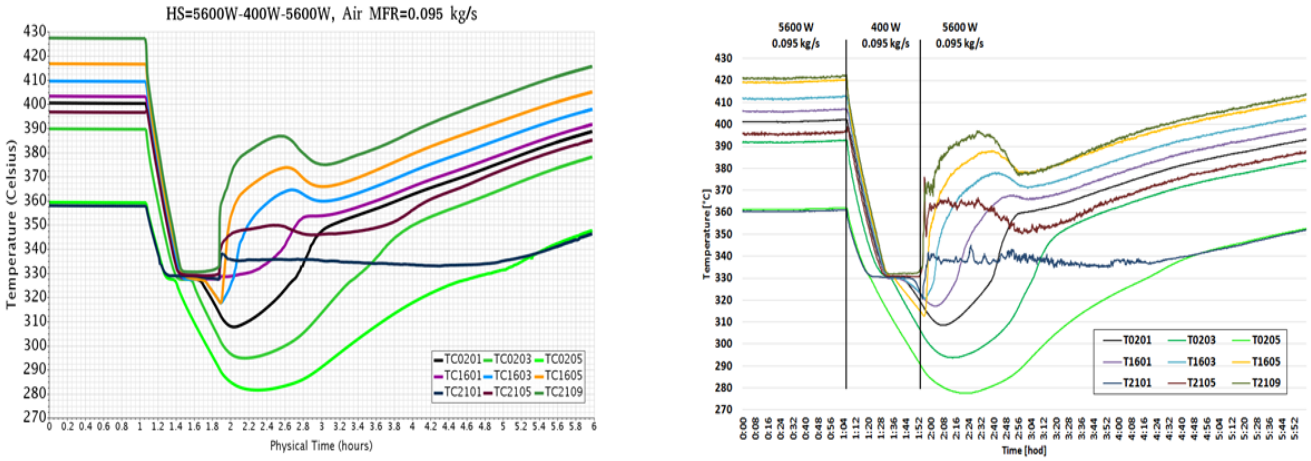



Fig. 6. Lead temperature on representative thermocouples in initial steady state (1 hour) followed by fast solidification (about 1 hour) and slow re-melting (about 4 hours) in the numerical model (left) and in the experimental test (right)

**CFD model of the ALFRED pool in SESAME**

In the framework of the SESAME WP3, the entire primary loop of the ALFRED reactor was modelled at CRS4 [11]. In order to get confident with the entire loop building and with the general design features, the construction of the first CFD model of ALFRED started from the design achieved in the LEADER project (illustrated in Fig. 7).

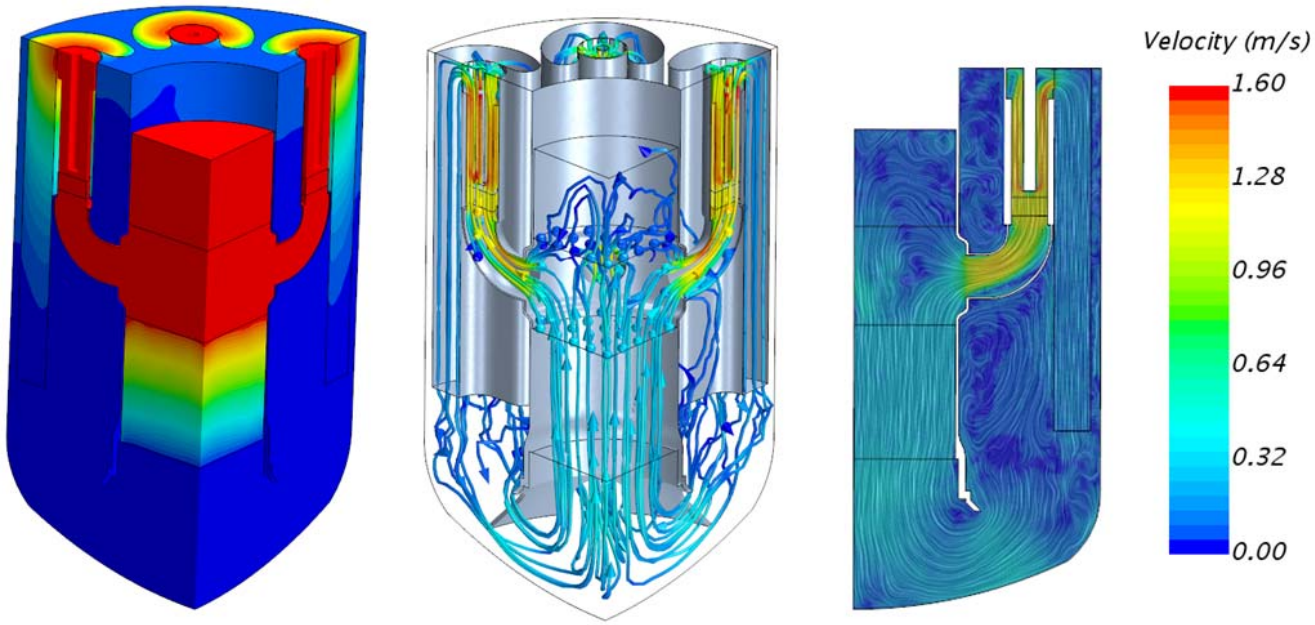



Fig. 7. General lead circulation in the loop of the ALFRED model according to the LEADER design

Furthermore, CRS4 was oriented to the simulation of various different configurations, from nominal operation to accidental operations. So, the ALFRED geometry was simplified to ease some degree of parameterization. This was in order to investigate swiftly several options based on a more recent ANSALDO concept only supported by a very simplified preliminary design.

The conductive heat transfer is however contemplated, making use of conducting baffle interfaces with thermal resistance defined in the correspondence of the solid walls. A modular approach was therefore adopted in the construction of the ALFRED geometry, allowing the simulation and analysis of a large number of configurations. Following the concept ideated by Ansaldo Nucleare, the cold pool is separated in two communicating regions with the cold lead from the heat exchanger circulating along the reactor vessel before turning to the core, to keep the external vessel as cold as possible and to minimize the thermal gradients in any condition. At this point of the analysis, only steady-states, many of them, are really needed. A strategy was developed in order to avoid the two-phase flow modelling. This is because two-phase flows require transient simulations, computationally much more demanding than steady-states. The computational domain was split in a sufficient number of sub-domains, to accommodate the model to the free surface levels of the different configurations: nominal (Fig. 8, left); Protected Loss of Flow (PLOF) (Fig. 8, centre), Loss of Coolant Accident (LOCA) (Fig. 8, right). The practical advantage of this approach is that the sub-domains can be activated or deactivated according to the simulated configuration, especially if more geometrical variants are investigated.

At first, the nominal steady state operation was simulated, reaching the full convergence in nominal configuration, followed by the representation of the PLOF and LOCA scenarios, in the context of a sensitivity study on the secondary cooling system and on the optimisation of the cold pools connection. The internal geometry is illustrated in Fig. 9 (left). The velocity and temperature fields on cross sections through the hot and cold pools intersecting the pump duct and the steam generator are shown in Fig. 9 (centre and right).

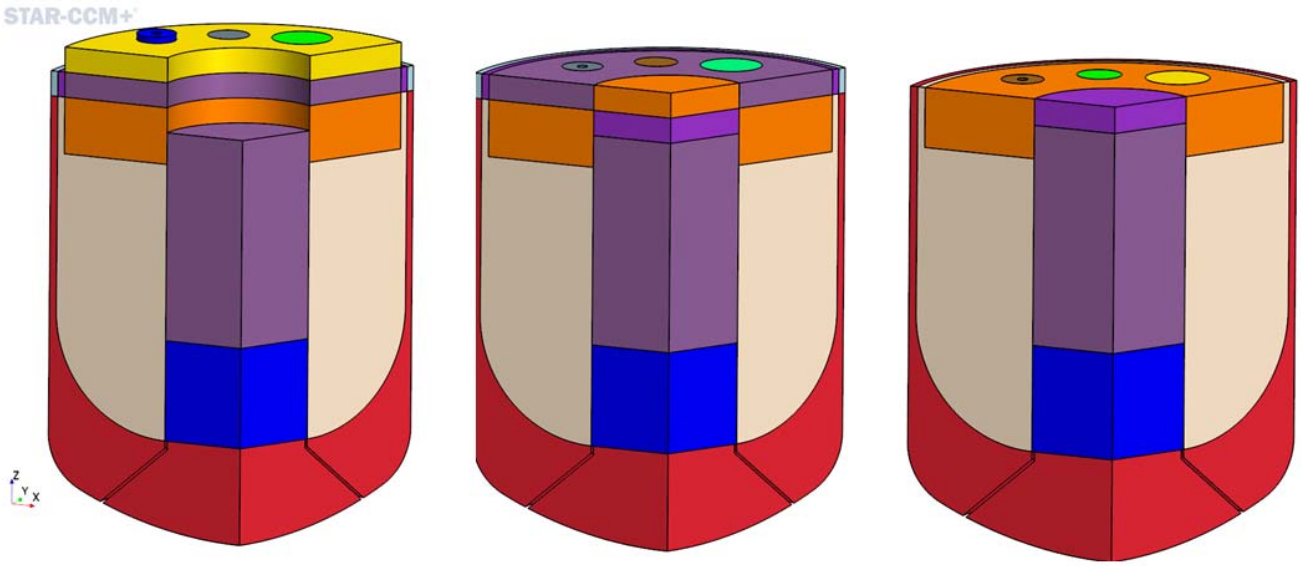



Fig. 8. Modular division of the computational domain: nominal (left), PLOF (centre), LOCA (right)

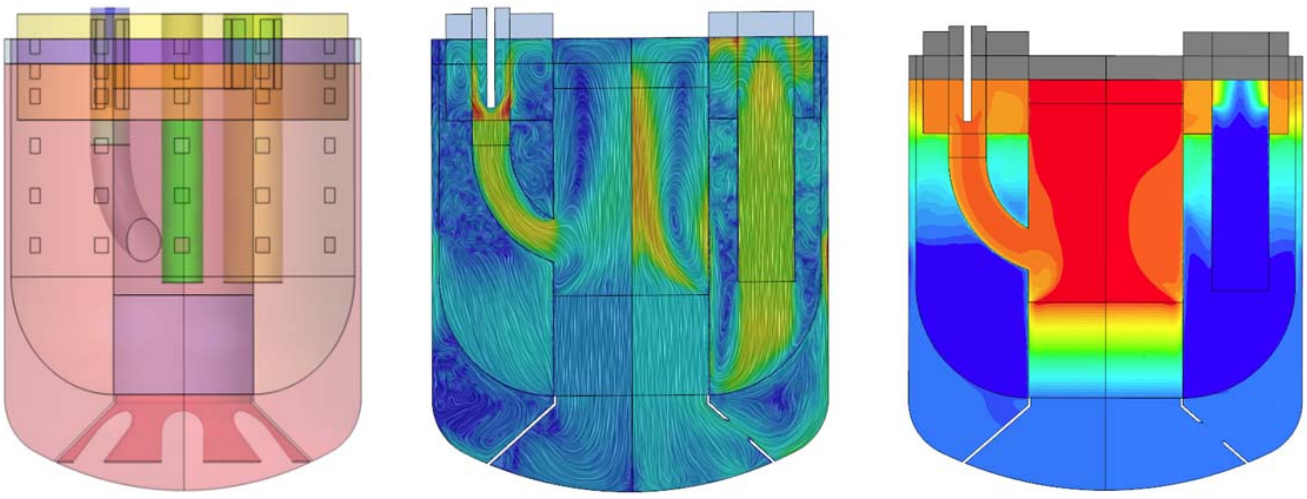


Fig. 9. CFD model of ALFRED according to the ANSALDO design: geometry (left), velocity field in PLOF scenario (centre), temperature field in PLOF+LOCA (right) scenario

**Concluding words**

Present paper described a complex and versatile CFD model of the MYRRHA primary coolant loop which reached a satisfying operation under nominal condition and behaved quite well under extreme postulated accidental scenarios, showing large safety margin.

The numerical modelling realized in the framework of the EURATOM projects in support to the development of the MYRRHA facility is giving information at intermediary scales which are, from one side, not accessible with the CFD analysis of single components and, from the other side, also not accessible to system codes. It is thus a complementary tool for the designer.

The correct modelling, most of the times, came step by step, aided by close collaboration with the experimentalists partners, who provided the measurements information (lead solidification experiment at CVR) or received important feedback from the numerical side with a consequently better understanding of the experiment (CR insertion experiment at SCK•CEN).

The control demonstrated on the CFD modelling has been reached thanks to the working continuity in the EURATOM projects. This allows CRS4 to propose itself as a complementary partner occupying its specific place in the network of specialists supporting the design of innovative GEN-IV nuclear systems. In particular, the acquired know-how definitively qualifies the CFD tool for the design support of the future Gen-IV ALFRED demonstrator.

**Acknowledgments**

The authors would like to thank the European Commission for its financial support within the FP7 SEARCH and MAXSIMA projects, H2020 SESAME and MYRTE projects and also the Autonomous Region of Sardinia.

**References**

[1] STAR-CCM+, http://mdx.plm.automation.siemens.com/star-ccm-plus

[2] Moreau V. (2009) ″A case history of CFD support to Accelerator Driven System Plant Design″, Proceedings of ICONE17, 75588, Brussels, Belgium

[3] Buckingham S. et al. (2015) ″Simulation of fuel dispersion in the MYRRHA-FASTEF primary coolant with CFD and SIMMER-IV″, Nuclear Engineering and Design, Vol. 295, pp. 74-83.

[4] Moreau V. et al. (2014) ″Challenges in the CFD modelling of MYRRA primary system″, Proceed. of the SEARCH/MAXSIMA Int. Workshop, Karlsruhe, Germany

[5] Moreau V. et al. (2019) ″An improved CFD model for a MYRRHA based primary coolant loop″, Nucl. Eng. and Design, Vol.353, https://doi.org/10.1016/j.nucengdes.2019.110221

[6] MAXSIMA project http://maxsima.sckcen.be/

[7] Profir, M., Moreau, V., Kennedy, G. (2017) ″Proof-Of-Principle of the MYRRHA Control Rod Emergency Insertion″, Journal of Nuclear Research and Development, No. 14, pp. 9-15

[8] Roelofs, F., Gerschenfeld, A., Tarantino, M., Van Tichelen, K., Pointer, W.D. (2016) ″Thermal-hydraulic challenges in liquid-metal-cooled-reactors″, Chapter in the book *Thermal Hydraulics Aspects of Liquid Metal Cooled Nuclear Reactors*, Woodhead Publishing Series in Energy.

[9] SESAME project http://sesame-h2020.eu/

[10] Profir, M., Moreau, V. (2019) ″Solidification model development″, SESAME project deliverable D3.13, https://cordis.europa.eu/project/rcn/198041/results/en

[11] Profir, M., Moreau, V., Alemberti, A., Frignani, M, Martelli, D. (2019) ″Pool CFD Modelling: Lessons from the SESAME Project″, Nucl. Eng. and Design, Vol. 355, https://doi.org/10.1016/ j.nucengdes.2019.110343