__DESCRIPTION OF THE RESEARCH PROJECT__

23.1. Scientific background, problem identification and objective of the proposed research

Scientific background

Heat transfer between solids and fluids (liquid or gas), or between different fluids, or between different phases of the same fluid, has been studied extensively for more than a century. For given fluids, the local heat transfer rate depends on physical quantities (pressure, temperature, fluid velocity, and others) as well as on the fluid conduits or enclosures (tubes, vessels). Besides, ranges of possible physical conditions are wide, possibly causing large variations of heat transfer rates that may strongly influence the behaviour of entire systems, not only quantitatively but also qualitatively. Given the possible wide range of both geometrical and physical conditions, heat transfer predicted in some situations by using existing models cannot always be considered reliable without proper validation based on experiments.

The proposed project deals with the specific case of heat transfer in small and modular nuclear reactors (SMRs) involving passive safety systems.

Problem identification

Although fundamental principles of heat transfer have been established, phenomena during some specific cases still cannot be modelled accurately. In particular, in the nuclear industry, heat transfer has been modelled for conventional nuclear power plants, whereas much smaller systems have mostly not been addressed yet, due to lack of necessity. Thus, it has not been sufficiently verified yet, whether models, developed for conventional plants, may be suitable for some conditions that may occur in smaller systems. In particular, the modelling of phenomena with a small driving force (a specific feature of passive systems) has proved to be challenging for computer codes used so far for simulations.

The development of SMRs in the past decade has resulted in many proposed concepts [1]. Even if we limit ourselves to light water(LW) SMRs, there are still many different options, so a specific design will not be singled out. However, despite these differences, passive safety systems are found in many LW SMRs, as they have the potential to offer a high level of safety by being able to ensure safety functions with a limited amount of electrical energy, system support and human actions [2-4]. The combination of low residual power and reliable passive system should lead to a significant reduction in the probability of core meltdown or radiological release to the environment. Among their safety functions, passive systems can remove heat from the reactor core, control the primary coolant inventory and maintain the integrity of the primary circuit. Thus, for successful and reliable simulation of accident mitigation in such SMRs, heat transfer models, implemented in computer codes, should be validated and eventually further developed. This refers both to descriptions on the local scale as well as on the system scale.

Objective of the proposed research

The objective of the proposed research is to model experiments representing parts of passive safety systems that are planned to be used in small and modular reactors (SMRs) to remove heat during an accident. The most critical gaps in modelling of heat transfer in passive safety systems will be identified, which should lead to further development and improvement. The modelling will be done both on the local scale as well as on the system scale. The developed models will be used in the future as building blocks of digital twins of developed SMRs.

On the system scale, state-of-the-art thermal-hydraulic system codes have been validated against expected operating reactor phenomena (observed in experiments). The validation should be extended to LW SMR passive systems phenomena based on representative experimental data.

On the local scale, the situation should be less uncertain, as models are based on either fundamental physical laws or universally accepted constitutive relations (such as Newton's law of viscosity or Fourier's law of heat conduction). Nevertheless, the integration of local conditions over entire systems may still result in inaccurate predictions, thus necessitating validation based on experimental results as well.

Among the phenomena involving heat transfer that were identified as the most challenging for modelling passive safety systems, the following will be considered;
- steam condensation in a conventional safety condenser tube,
- reactor pressure vessel external cooling,
- gravity-driven injection from accumulators.

23.2. State-of-the-art in the proposed field of research and survey of the relevant literature
(see list of references at the end)

As the three considered phenomena have been extensively investigated, both as basic phenomena as well as phenomena that occur in large systems, a comprehensive state-of-the-art and survey of all the relevant literature would be beyond the scope of the proposed project. For this reason, the review is limited to safety systems.

Steam condensation in a conventional safety condenser tube

Condensation in a vertical tube is very often considered in studies of heat removal systems. However, most works (at least those that deal with such systems as part of safety systems) deal only with entire heat exchangers [5-8]. Nevertheless, some passive-safety-systems-related works deal also with the detailed analysis of condensation in the tube and heat transfer on the tube outer wall (specifically, with the effects of tube inclination) [9-10]. Thus, works that would consider condensation in detail (be it on the system or local scale) in the specific case of safety systems are lacking. No reference was found dealing with the specific case of condensation in a safety condenser tube in some proposed SMR design.

External cooling of reactor pressure vessel

The issue of external cooling of the reactor pressure vessel lower plenum has been investigated at least for some 30 years. Relatively recent theoretical works are described in refs. [11-18]. However, all these works (and others found in the literature) deal with conditions typical for conventional size nuclear power plants. No reference was found dealing with external cooling of a reactor pressure vessel in some proposed SMR design.

Gravity-driven injection from accumulators

The development of new accumulator concepts does not consist in revolutionary new designs but of gradual modifications. However, the proposed advances are related to conventional nuclear power plants [19-24], and none of them proposes gravity-driven injection. Also, none of the works found in the literature deals explicitly with a new accumulator concept to be applied to SMRs.

23.3. Detailed description of the work programme

The experiments that represent parts of passive safety systems will be modelled using the following types of computer codes:

- Computational Fluid Dynamics (CFD) codes that use local instantaneous description of the fluid, be it in conduits or vessels or compartments. These simulations will enable the application of the codes to some parts of SMRs where highly heterogeneous or multi-dimensional phenomena could be expected. Presently, the use of the open-source CFD code OpenFoam is foreseen.

- System computer codes that use one-dimensional instantaneous description of the flow in conduits and volume-averaged descriptions of fluids in large vessels or compartments. These simulations will enable the codes to be applied when simulating accidents in entire SMRs. Presently, the use of the thermal-hydraulic system code RELAP5 is foreseen.

Experimental results will be available through the participation of the proposing research organisation in the European project "Ensuring Assessment of Safety Innovations for SMR" (EASI-SMR, 2024-2028), coordinated by Electricité de France. The participation in the EASI-SMR project is not only qualitatively smaller than the proposed work (only simulations of condensation and RPV external cooling with a CFD code will be performed), but also quantitatively (more than five times smaller) and represents only the basis for the presently proposed project.

The specific work programmes for the phenomena, listed at the end of section 23.1., are now presented.

Steam condensation in a classical safety condenser tube

A passive system for heat removal may involve a two-phase natural circulation loop featuring a safety condenser-type heat exchanger submerged in a pool. Heat exchange in the condenser tubes involves both condensation and cooling heat flow on the two-phase circulation loop side, and heating heat flow (and boiling) on the heat sink side. Heat exchange by boiling in the outer tube wall is extremely efficient, but natural circulation in a volume is complex to model and reproduce by thermal-hydraulic codes, and two-phase thermal-hydraulic CFD codes are not sufficiently validated for such systems. Uncertainties in film condensation models, heat exchange between vapor and liquid phases (film on tube inner wall), and convective liquid exchange in the pool leads to a final uncertainty in the modelling of the two-phase natural circulation loop. It is therefore necessary to separate the physical parameters characterizing condensation and cooling, on the one hand, from the physical parameters and models characterizing boiling and overall heating of the pool, on the other.

Two sets of experiments, performed in the SACO facility (Framatome GmbH, Germany), shown schematically in Figure 1, will be simulated to study the steam condensation at passive conditions in single tubes with dimensions characteristic of those of a safety condenser:

- Experiments covering the full range of steam pressures and temperatures likely to be generated in an SMR safety loop for the most referent deterministic transient design basis accidents. This experimental campaign will cover a series of set points, in a large range of saturated steam pressure and temperature, for the comparison of heat transfer flux between thermal-hydraulic system code simulation and experimental measurements. These experiments will be performed in the COSAC condensation tube.

- Experiments that will provide complementary high-resolution data to the previously mentioned experiments, for a limited range of steam pressures and temperatures for validation of system and CFD codes related to film condensation, with and without the presence of non-condensable gases. These experiments will be performed in the PRECISE condensation tube (to be provided by the Paul Scherrer Institute, Switzerland).

Detailed specifics of the condensation experiments are currently being planned within the European EASI-SMR project.

Reactor pressure vessel external cooling

An experimental program has been set at UJV Rez (Czechia) with the main purpose to investigate specific physical behaviour of cooling systems during in-vessel retention (IVR) conditions (that is, the conditions when the reactor pressure vessel (RPV) remains whole and the reactor core melt remains in the vessel lower plenum), for pool boiling and thermosiphon configuration. A schematic of the experiment is shown in Figure 2.

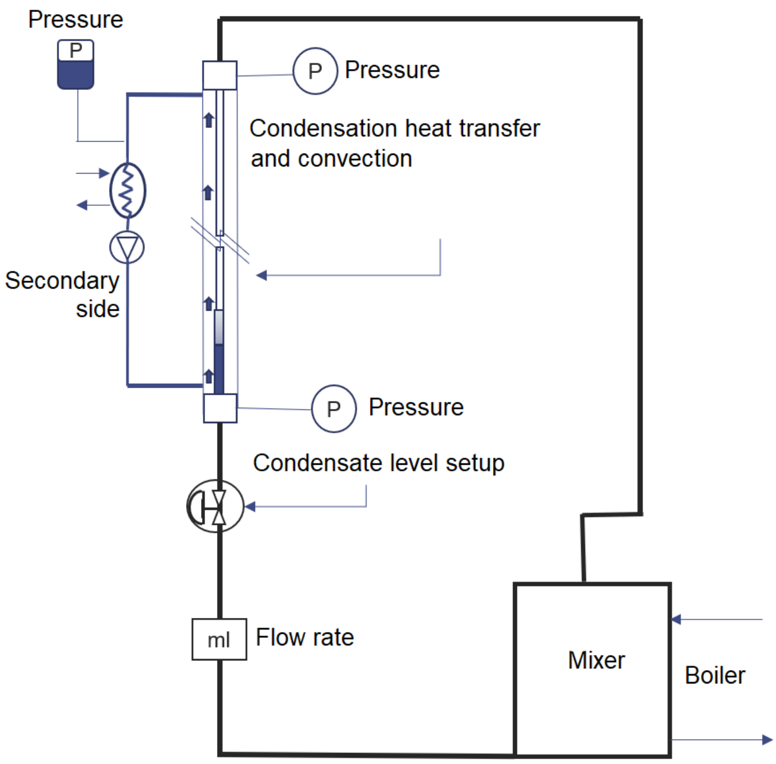

Figure 1. Schematic of SACO experimental setup
to study steam condensation in a vertical tube.

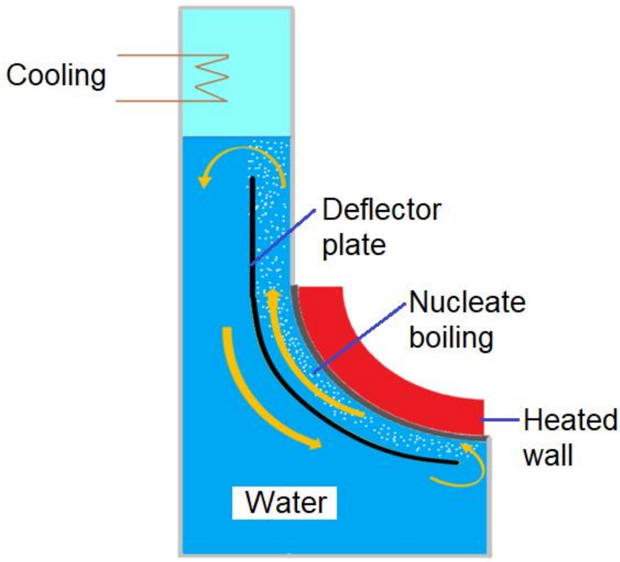

Figure 2. Experimental setup for investigation of reactor pressure vessel
external cooling during in-vessel melt retention.

Possible boundary conditions to be set for this experimental program are:
- heat flux profile along reactor pressure vessel outer surface;
- geometry of deflector plates (distance form RPV, overall height, inlet part geometry);

- parameters of coolant – pressure range, temperature;
- water level inside experimental channel.

Detailed specifics of the cooling experiments are currently being planned within the European EASI-SMR project.

The purpose of the simulation will be to simulate the cooling rate of the reactor pressure vessel wall element, determined in the experiment. The emphasis will be on the simulation with a CFD code, using two-dimensional modelling. However, a system code will also be used to investigate the possibility to simulate an accident in an entire SMR.

Gravity-driven injection from accumulators

During a loss-of-coolant accident, the primary water inventory decreases and safety injection systems are usually actuated to maintain sufficient level in the vessel reactor, able to ensure a flooded reactor core. In some SMR designs, such passive systems are not classic accumulators with a pre-pressurization of nitrogen, commonly known and used in PWR concepts, but gravity draining accumulators (although this concept is actually also used in some PWRs). The interest of such passive system (that is, gravity-driven) is to inject water during a loss-of-coolant transient with no dependence on the depressurization rate of the primary circuit.

The GRADAC experiment (Lappeenranta University of Technology, Finland), schematically shown in Figure 3, consists in reproducing and thermo-hydraulically modelling the behaviour of an emergency injection system in the event of a loss-of-coolant-type accident. The experiment that will address the primary inventory control consists in reproducing the initial pressurization phase of the primary system until equilibrium is reached. Two tanks represent the reactor vessel and the accumulator, and two connecting lines will permit pressurization of the accumulator and injection in the vessel. The condensation of injected steam on the cold free liquid surface will also be studied and modelled, as will the thermal absorption of the upper metal walls.

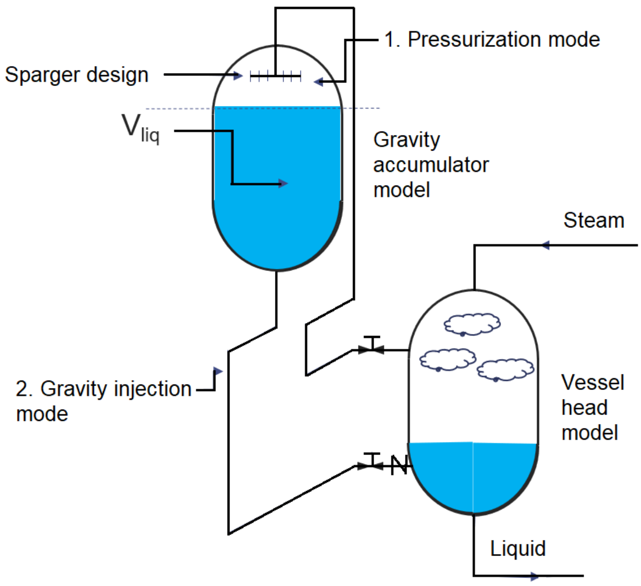

Figure 3. Schematic of GRADAC experiment
on gravity-driven accumulator injection.

Four tests are planned, with the first one to be carried out with an unpressurized cold facility. The aim of the experiment is to test how the gravity-driven accumulator injects water. Different sparger designs will be tested with three experiments to optimize steam distribution in the accumulator plenum, to minimize the direct impact of the steam jet on the liquid surface, and to cause surface disturbances likely to break up the upper warmer liquid layer that reduces the effect of free-surface condensation.

Detailed specifics of the gravity-driven injection experiments are currently being planned within the European EASI-SMR project.

The purpose of simulations will be to assess the capability of system codes to simulate the gravity-driven accumulator injection in actual SMRs. The simulation of the entire system over a longer period is beyond the capabilities of CFD codes. However, the liquid surface disturbances in the accumulator will be separately simulated with a CFD code to investigate the breaking up of the warmer liquid layer. The obtained results might then be incorporated in the modelling with the system code.

23.4. Available research equipment over 5.000 €

High Performance Computer cluster with a total of 5500 computing cores (25 TB RAM, 900 TB disk space), co-owned by research teams of Reactor Engineering Division and Reactor Physics Division of the Jožef Stefan Institute. Fifteen (15) nodes available with GPGPU architecture.

Installed (and used) software for Computational Fluid Dynamics analyses:

- Open source CFD codes: PSI-Boil (Paul Scherrer Institute, Switzerland), OpenFoam and Code-Saturne (Electricité de France);
- Commercial CFD suite ANSYS (CFX, Fluent).

23.5. Project management: Detailed implementation plan and timetable

The detailed implementation is related to the schedule of the European project EASI-SMR through which the experimental results will be available. However, preliminary models of experimental facilities will be developed based on already available data.

The planned work in all considered topics will (in general, except in some specific cases) follow the same pattern:

- development of preliminary model from already available data,
- preliminary simulations assuming generic initial and boundary conditions (the term "generic" designates ad-hoc conditions that are within the ranges of similar experiments, described in the literature),
- refinement of model based on data received with experimental results,
- simulations applying experimental initial and boundary conditions,
- comparison between experimental and simulation results,
- analysis of possible modelling deficiencies and eventual modelling improvement.

Steam condensation in a conventional safety condenser tube

The specificity of this topic is that simulations with a thermal-hydraulic system code will be performed for both considered tubes (COSAC and PRECISE), whereas simulations with the CFD code will be performed only for the tube for which detailed measurements will be available (PRECISE). Preliminary simulations will be performed only with the system code. In addition to the already stated purposes, the dual simulation of experiments in the PRECISE tube will be an opportunity for a direct comparison of capabilities of system and CFD codes.

Task	Period (months)
• Development of preliminary COSAC tube model for system code • Preliminary simulation of condensation in COSAC tube using system code with prescribed generic conditions	M1-M3
• Development of preliminary PRECISE tube model for system code • Preliminary simulation of condensation in PRECISE tube using system code with prescribed generic conditions	M4-M6
• Refining of COSAC tube model for system code and simulation of performed experiments • Comparison between experimental and simulation results • Analysis of possible modelling deficiencies and eventual model improvement	M7-M9
• Refining of PRECISE tube model for system code and simulation of performed experiments • Comparison between experimental and simulation results • Analysis of possible modelling deficiencies and eventual model improvement	M10-M12
• Development of PRECISE tube model for CFD code • Simulation of condensation experiments in PRECISE tube using CFD code	M13-M18
• Analysis of possible deficiencies of PRECISE experiment modelling with CFD code and eventual model improvement	M19-M24
• Comparison of all simulation results with other results from EASI-SMR project	M31-M36

Responsible researchers: Matej Tekavčič, Iztok Tiselj

Reactor pressure vessel external cooling

The specificity of this topic is that simulations with a CFD code will be performed first. Namely, the simplified descriptions of system codes are much more suitable to the phenomena in tubes than to phenomena in vessels. Thus, simulations of experiments on heat transfer on the wall, immersed in a pool, with the system code will be performed after getting familiar with the phenomena, based on CFD simulations. Preliminary simulations will be performed only with the CFD code. As for condensation, the dual simulation will be an opportunity for a direct comparison of capabilities of system and CFD codes.

Task	Period (months)
• Development of preliminary model of IVR wall facility for CFD code • Preliminary simulation of boiling on IVR facility wall surface using CFD code with prescribed generic conditions	M7-M12
• Refinement of IVR facility model for CFD code and simulation of performed experiments • Comparison between experimental and simulation results • Analysis of possible modelling deficiencies and eventual model improvement	M13-M18
• Development of model of IVR wall facility for system code • Simulation of performed experiment in IVR wall facility using system code • Comparison between experimental and simulation results	M19-M24
• Comparison of all simulation results with other results from EASI-SMR project	M31-M36

Responsible researchers: Janez Kokalj, Ivo Kljenak

Gravity-driven injection from accumulators

The specificity of this topic is that the system is much too large for simulations on a local scale (using the CFD code). On the other hand, simulations of the liquid surface disturbances that may break up the upper warmer liquid layer should be performed on the local scale (using the CFD code) but cannot be directly simulated with the system code. However, the possible modelling of the influence of the disturbances in the input model for the system code will be investigated.

Task	Period (months)
• Development of preliminary model of GRADAC facility for system code • Preliminary simulation of GRADAC experiment with prescribed generic conditions	M13-M18
• Refinement of GRADAC model for system code and simulation of performed experiments • Comparison between experimental and simulation results • Analysis of possible modelling deficiencies and eventual model improvement	M19-M24
• Development of liquid surface model for CFD code • Simulations of steam impact, surface disturbance and breakup of warmer liquid layer with CFD code	M25-M30
• Investigation of possible modelling in system code to consider the effects of surface disturbances	M31-M36
• Comparison of all simulation results with other results from EASI-SMR project	M31-M36

Responsible researchers: Andrej Prošek, Leon Cizelj

REFERENCES

General

[1] OECD NEA, The NEA Small Modular Reactor Dashboard: Second Edition, NEA No. 7671, 2024.

[2] Butt H.N., Ilyas M., Ahmad M., Aydogan F., Assessment of passive safety system of a Small Modular Reactor (SMR), Annals of Nuclear Energy 98 (2016) 191–199

[3] Jeon B.G., Bae H., Kim S.H., Yang J.H., Ryu S.U., Yi S.J., Park H.S., Experimental and analytical investigation on SMART passive safety systems for three 2-inch SBLOCA tests using SMART-ITL, Annals of Nuclear Energy 188 (2023) 109835

[4] Mascari F., Woods B.G., Welter K., D'Auria F., Bersano A., Maccari P., Small modular reactors and insights on passive mitigation strategy modeling, Nuclear Engineering and Design 401 (2023) 112088

Modeling of steam condensation in a classical safety condenser tube

[5] Chang S.H., Kim S.H., Choi J.Y., Design of integrated passive safety system (IPSS) for ultimate passive safety of nuclear power plants, Nuclear Engineering and Design 260 (2013) 104– 120

[6] Yang Z., Shan J., Gou J., Preliminary assessment of a combined passive safety system for typical 3-loop PWR CPR1000, Nuclear Engineering and Design 313 (2017) 148–161

[7] Park S.D., Lee D.W., Kang K.J., Park H.-S., Numerical study on the thermal-hydraulic behavior in the ultimate heat sink of passive residual heat removal system in the SMART, Nuclear Engineering and Design 399 (2022) 111997

[8] Chang W., Xiaohui L., Jiaming C., Guanghuai W., Experimental investigation on characteristics of Multiple-Loops coupled natural circulation system, Nuclear Engineering and Design 407 (2023) 112283

[9] Minocha N., Joshi J.B., Nayak A.K., Vijayan P.K., 3D CFD simulations to study the effect of inclination of condenser tube on natural convection and thermal stratification in a passive decay heat removal system, Nuclear Engineering and Design 305 (2016) 582–603

[10] Amidu M.A., Park J., Park Y., Lim C., Jung S., Kim H., Performance analyses of a steam condensation tube immersed in a saturated water pool: Effects of tube inclination, Nuclear Engineering and Design 323 (2017) 142–155

Reactor pressure vessel external cooling

[11] Tusheva P., Altstadt E., Willschütz H.-G., Fridman E., Weiss F.-P., Investigations on in-vessel melt retention by external cooling for a generic VVER-1000 reactor, Annals of Nuclear Energy 75 (2015) 249–260

[12] Whang S., Park H.S., Moriyama K., Lim K., Cho Y.J., Kim M.H., Uncertainty analysis of in-vessel retention in a high power reactor during severe accident, Nuclear Engineering and Design 319 (2017) 1–11

[13] Hu Q., Yan X., Huang S., Yu J., The comprehensive analysis of coolability limits of passive external reactor vessel cooling under in-vessel retention, Annals of Nuclear Energy 120 (2018) 296–303

[14] Amidu M.A., Addad Y., Lee J.I., Kam D.H., Jeong Y.H., Investigation of the pressure vessel lower head potential failure under IVR-ERVC condition during a severe accident scenario in APR1400 reactors, Nuclear Engineering and Design 376 (2021) 111107

[15] Colombo M., Fairweather M., Study of nuclear reactor external vessel passive cooling using computational fluid dynamics, Nuclear Engineering and Design 378 (2021) 111186

[16] Pop A., Petruzzi A., Giannotti W., External function for GOTHIC code to estimate critical heat flux conditions for in-vessel retention assessment, Nuclear Engineering and Design 380 (2021) 111301

[17] Akand M.A.R. , Matsumoto T., Liu W., Morita K., Mechanistic critical heat flux prediction for in-vessel retention conditions, Nuclear Engineering and Design 384 (2021) 111494

[18] Rababah S., Diab A., Evaluation of a hybrid in-vessel retention strategy with ex-vessel cooling for APR1400 under extended station blackout conditions, Nuclear Engineering and Design 429 (2024) 113600

Gravity-driven injection from accumulators

[19] Shiraishi T., Watakabe H., Development of the advanced accumulator for the pressurized water reactor, Nuclear Engineering and Design 249 (2012) 318– 334

[20] Liu X.J., Fu S.W., Xu Z.H., Yang Y.H., Cheng X., LOCA analysis of SCWR-M with passive safety system, Nuclear Engineering and Design 259 (2013) 187– 197

[21] Hu H., Shan J., Gou J., Cao J., Shen Y., Fu X., Simulation of advanced accumulator and its application in CPR1000 LB LOCA analysis, Annals of Nuclear Energy 69 (2014) 183–195

[22] Pouresgandar M., Safarzadeh O., Talebi S., Evaluation of advanced accumulator in a VVER-1000 reactor in loss of coolant accident, Annals of Nuclear Energy 170 (2022) 108988

[23] Cadiou T., Stratta E., Augier L., Multi-Scale study of an innovative safety system for pressurized water reactors, Nuclear Engineering and Design 387 (2022) 111600

[24] Redondo-Valero E., Queral C., Fernandez-Cosials K., Sanchez-Espinoza V., Safety margins improvement by means of the passive second stage hydroaccumulators in a VVER-1000/V320 reactor, Nuclear Engineering and Design 414 (2023) 112644