

Coupling of Computational Fluid Dynamics Codes with Structural Mechanics and System Thermal-Hydraulics Programs and Their Validation on Nuclear Reactor Relevant Experiments

A. Papukchiev^{*}

Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Boltzmannstr. 14, 85748 Garching n. Munich, Germany

Abstract. Nowadays System Thermal Hydraulics and computational fluid dynamics programs are being coupled to provide numerical solution for the simulation of nuclear reactor transients and accidents with the consideration of 3D flow and heat transfer effects, which are phenomenologically accurate and affordable in terms of CPU time. In the past years, GRS developed a coupling algorithm for the 1D STH code ATHLET and the 3D CFD tools ANSYS CFX and OpenFOAM. Within several projects, the multiscale program ATHLET-ANSYS CFX was verified and validated on different experiments like LSTF ROSA V Test 1.1, ROCOM, TALL-3D, etc. For the evaluation of reactor relevant phenomena including interaction between fluids and solids such as flow-induced vibrations, CFD is being coupled to computational structural mechanics codes. At GRS, ANSYS codes were applied and validated against different FIV experiments in nuclear power plants. Some of the results from the multiscale and the multiphysics validation cases are briefly summarized in this paper.

Keywords: multiscale, multiphysics, coupling, CFD, CSM, ATHLET, ANSYS, flow-induced vibrations

1 Introduction

System Thermal Hydraulics (STH) codes have been developed and applied in the last 50 years to model the behavior of nuclear power reactors (NPP). These provide reliable results at relatively low computational cost. Being based on 1D lumped parameter approach, System Thermal Hydraulics codes (STH) show deficiencies in transients and accidents with complex 3D flow phenomena such as mixing, stratification or strong flow asymmetry in large fluid volumes. Hence, initiatives in the nuclear community promote the utilization of computational fluid dynamics (CFD) programs for the evaluation of specific reactor safety issues. The CFD codes were developed for an accurate and detailed solution of fluid flow in three dimensions. Unfortunately, the representation of a complete reactor cooling circuit is still impractical due to the very high computational time. Therefore, STH and CFD codes are coupled for a computationally more affordable solution with high resolution in particular areas of interest. In the past years, GRS developed a coupling algorithm for the 1D STH code ATHLET and the 3D CFD tools OpenFOAM and ANSYS CFX. The ATHLET-CFD coupling has been verified and validated on several experiments related to the current and next generation nuclear systems.

One validation calculation was performed on the Large Scale Test Facility (LSTF) ROSA V Test 1.1, carried out within the OECD/NEA Rig Of Safety Assessment project in Japan [1]. It is dedicated to Pressurized Thermal Shock (PTS) issue and deals with flow mixing and temperature stratification during emergency coolant injection in the primary circuit of a pressurized water reactor (PWR) at natural circulation conditions. LSTF is a full height,

1/48 volume-scaled facility that represents a Westinghouse PWR. Further validation was done on several ROCOM experiments. ROCOM is a test facility replicating German KONVOI (Siemens KWU) type PWR. In the ROCOM PKLIIT1.1 experiment, ethanol is injected into the water-filled ROCOM circuit, leading to complex flow mixing phenomena in the facility [2]. Within the European THINS and SESAME projects, ATHLET-CFD was extended for the simulation of Generation IV reactors. The TALL-3D thermal hydraulic loop is a three-leg, 7 m high integral experimental facility, equipped with one main heater and one heated test section (TS), positioned in two different legs [3]. Complex thermal-hydraulic phenomena including mixing and stratification of the lead-bismuth eutectic coolant occur in the TS. The experimental data by the Swedish researchers was used to validate the coupled code for liquid metal flows.

Under certain conditions, flow-induced vibrations (FIV) might develop in steam generators, the reactor core, or other NPP components. These phenomena could potentially damage some of the barriers for safe confinement of core inventory, and therefore, need to be carefully analyzed. To provide an accurate and reliable analysis of FIV, today, modern 3D CFD and computational structural mechanics (CSM) codes are coupled to predict the interaction between fluids and solids. The multiphysics codes ANSYS CFX-MOR and ANSYS CFX-Mechanical were applied and validated by GRS against the Vattenfall Neutron Detection Housing Vibration Experiment [4]. This was dedicated to the vibration of instrumentation guide tube in the core of a boiling water reactor (BWR). GRS participated in the international OECD/NEA FSI Benchmark that took place in

¹Corresponding author e-mail: angel.papukchiev@grs.de

2021-2022. In this work, some of the achieved results on the vibration of two in-line cylinders, exposed to a cross-flow in a rectangular flow channel, are described [5].

The objective of this paper is to briefly summarize the efforts at GRS to validate multiscale and multiphysics programs. Focus is put only on the main outcome from selected validation cases and their analysis. For the interested reader, references to detailed description of the work performed and the results achieved are provided for each case. In the end, conclusions are drawn and outlook for future work is provided.

2 Multiscale STH – CFD Coupling

Regarding the coupling in space, STH and CFD codes are coupled using domain decomposition or domain overlapping strategies. In domain decomposition the whole computational domain is split in subdomains, and each one of these is simulated either by the STH or by the CFD code. At the subdomain boundaries, thermal-hydraulic (TH) data are exchanged via coupling interfaces. In domain overlapping strategies the whole geometry is calculated by the STH code, while the regions with pronounced 3D flow phenomena are treated with the CFD program. The CFD solution is then used to “correct” the STH results in these regions. The ATHLET – CFD coupling approach is based on domain decomposition strategy [6]. Figure 1 shows a simple open TH system, consisting of two ATHLET and one ANSYS CFX pipe. In hydraulically coupled codes, one of the programs provides scalar variables (pressure, fluid temperature, quality, etc.) and the other calculates vector variables (mass flow or velocity and related temperature, etc.) at the coupling interface. Regarding the coupling in time, explicit and semi-implicit coupling schemes have been developed [7].

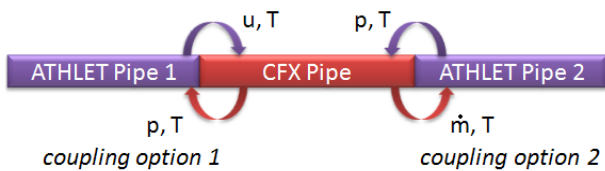


Figure 1. Multiscale coupling: thermal-hydraulic exchange parameters for single-phase flow problems.

3 Validation of ATHLET-ANSYS CFX on the LSTF ROSA V Test 1.1 Experiment

3.1 Description of the LSTF facility and boundary conditions of the ROSA V Test 1.1 experiment

The Japanese Large Scale Test Facility (Figure 2) represents a four-loop, 3423 MW thermal power Westinghouse PWR by a full-height and 1/48 volumetrically scaled two-loop (named loop A and loop B) system. The cold legs are similar and consist of straight and elbow parts which are attached to the main coolant pumps. However, the emergency core cooling (ECC) injection nozzles in the cold legs differ in shape and location. ECC Nozzle A is perpendicular to the main pipe simulating VVER reactor type conditions, while ECC Nozzle B forms a 45° angle with the cold leg, simulating a Western type PWR. The ROSA V Test 1.1 was

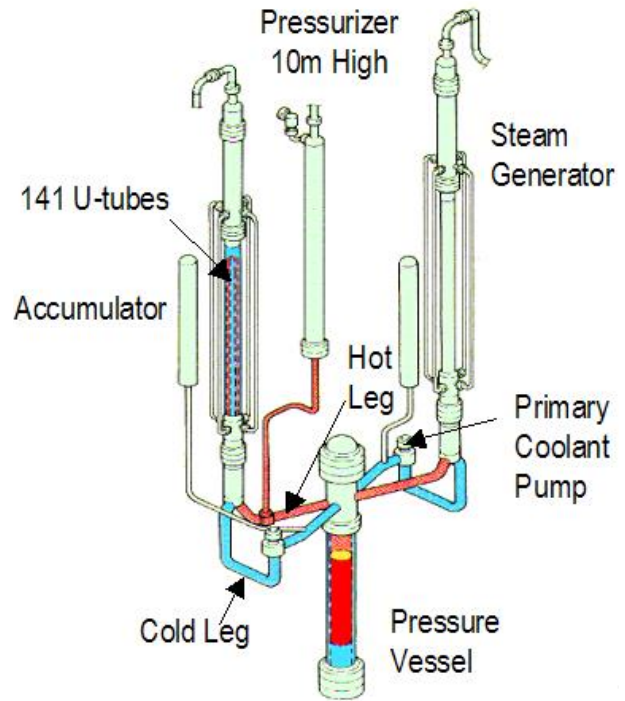


Figure 2. Large Scale Test Facility in Japan [1].

performed in 2006 within the framework of OECD/NEA ROSA project and was dedicated to the Pressurized Thermal Shock (PTS) scenario [1]. The goal of the experiment was to investigate flow mixing and temperature stratification after a cold (~35°C) emergency core cooling (ECC) injection in a hot water filled primary circuit (280°C) under natural circulation conditions, and to provide data for the validation of computer codes. Temperatures were measured with thermocouple (TC) rakes in the cold leg A below the injection nozzle, and at two cross-sectional planes between the injection nozzle and the downcomer. Each rake in the cold leg consists of 21 TCs, positioned in three columns and seven rows. Additionally, 18 TCs were installed in the downcomer below the cold legs.

ROSA V Test 1.1 started at forced circulation, and when the pumps were switched off. Natural circulation at 15.5 MPa and 2% core power established in the primary LSTF circuit. The simulation results presented in this paper are focused on the first phase of Test 1.1, where ECC water was injected for about 110 s into the cold leg A at these conditions. The temperature difference between the hot water in the primary system and the cold ECC water was almost 250 K, resulting in a density difference of more than 200 kg/m³.

3.2 Large Scale Test Facility modeling with ATHLET-ANSYS CFX

For the 1D system code calculations, the existing detailed ATHLET input deck for the LSTF was used. Both loop A and loop B were modeled and connected to an RPV with two-channel core and downcomer representation. A fine nodalization scheme with three different heat exchanger U-tube bundles (short, medium and long) has been developed for the modeling of the steam generators. For the CFD part of the coupled simulation a CAD model of the LSTF cold legs, the ECC lines, and the downcomer was

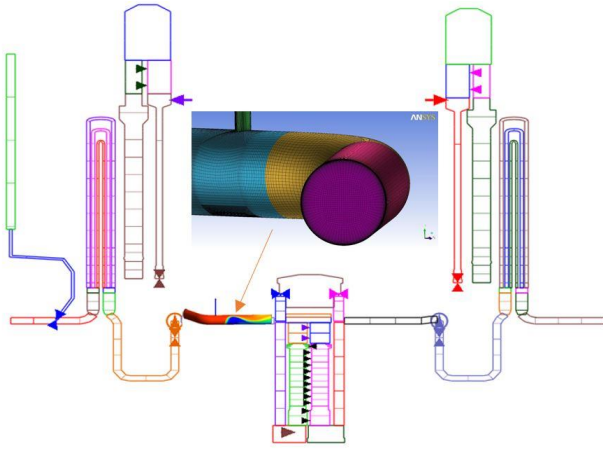


Figure 3. Coupled ATHLET-ANSYS CFX model of the LSTF facility.

generated on the basis of the available drawings. In this study, only the part of the LSTF cold leg A between the main coolant pump and the reactor pressure vessel (RPV) downcomer, which includes the ECC injection nozzle, was modeled. A grid with 1.3 million elements has been selected for the final 3D simulations. The BSL RSM turbulence model was selected for all calculations and a constant time step size of $5E-2$ s has been selected for both ANSYS CFX stand-alone and the coupled ATHLET – ANSYS CFX calculations. In a next step, the coupled 1D-3D LSTF model was developed. The 1D ATHLET pipe section in the cold leg A between the main coolant pump and the downcomer was “replaced” by a 3D ANSYS CFX pipe section. The ATHLET – ANSYS CFX model of the LSTF can be seen in Figure 3. The part of the primary circuit with relevant 3D effects is treated with ANSYS CFX, and ATHLET is used to provide fast solution for the flow behavior in those regions where 1D simulation is adequate.

3.3 Analysis and validation results for LSTF ROSA V Test 1.1

The transient was simulated with ATHLET-ANSYS CFX and ANSYS CFX stand-alone. In the CFD stand-alone simulations, time-dependent, measured mass flow rates and temperatures were specified at the inlet boundaries. In the coupled 1D-3D calculations, the transient mass flow rates and temperatures were calculated by ATHLET – ANSYS CFX. Figure 4 shows the calculated temperature distribution during the ECC injection at the wall of the cold

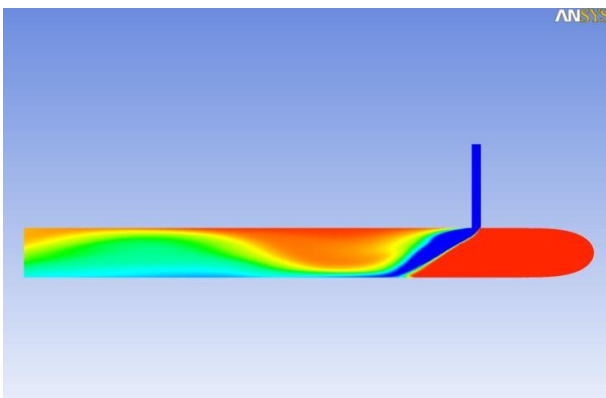


Figure 4. Temperature distribution in the LSTF cold leg A.

leg A approx. 100 s after transient begin. The vertically downwards injected cold ECC water impinges on the cold leg bottom and then sloshes to the left and right pipe walls. Due to its higher density, the cold water pushes the lighter hot water to the top and gradually stratifies at the bottom of the cold leg. Horizontally shaped and stratified temperature layers can be observed at approx. 0.3 m distance from the RPV entrance.

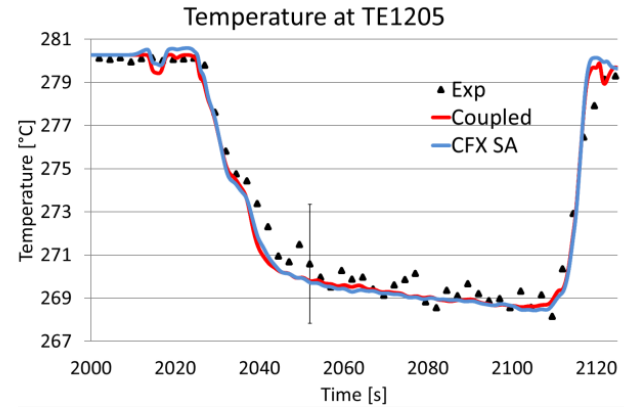


Figure 5. Temperature progression at TE1205 thermocouple.

Figure 5 shows the comparison for TE1205, which is positioned at the bottom of the cold leg A and close to the RPV entrance. It can be seen that both, ANSYS CFX stand alone and coupled ATHLET-ANSYS CFX results, agree very well with data. During the transient the TE1205 temperature decreases by approx. 12 K, while the temperature in the upper part of the pipe cross section decreases by only 2-3 K. The temperature drop by 12 K at the bottom of the cold leg A is not small. Its impact on the structural integrity of the downcomer wall has to be analyzed with the help of a multiphysics CFD-CSM programs. Further, the very good agreement between ANSYS CFX stand-alone and ATHLET – ANSYS CFX calculations proves the consistency of the ATHLET – ANSYS CFX coupling methodology for single-phase flows. Small differences between ANSYS CFX and ATHLET – ANSYS CFX temperatures are due to the different boundary conditions in the simulations. In the ANSYS CFX calculation these are measured data, while in the coupled code calculation, the cold leg A mass flow rate and temperature result from the coupled ATHLET – ANSYS CFX solution. Detailed description of this analysis can be found in [8].

4 Validation of ATHLET-ANSYS CFX on the ROCOM PKLIII T1.1 Experiment

4.1 Description of the ROCOM facility and the PKLIII T1.1 experiment

ROCOM is a test facility replicating German KONVOI (Siemens KWU) type pressurized water reactor (PWR). All important details up to the core inlet are represented at a scale of 1:5. Water at room temperature and atmospheric pressure is used to determine the coolant mixing behavior. The water mass flow rate in the loops is controlled by pumps, offering the possibility to operate the facility at wide range of flow conditions. Salt or brine tracer additives are used to alter the local electrical conductivity of

the fluid. The tracer labels specific volume of water and then its distribution within the system is measured. This is monitored by the specially designed wire-mesh electrical conductivity sensors (WMS) that provide high-resolution measurements of the tracer concentration in space and time. The ROCOM PKLIIT1.1 experiment [2] is dedicated to the assessment of natural circulation interruption phenomenon that depends on the balance of the buoyancy driving forces in the primary circuit and the pressure loss in it. In PKLIIT1.1 the volumetric flow rates in loops 1, 2 and 4 are kept constant at their nominal values of 3.3 l/s (0.188 m/s) throughout the whole experiment. The ECC injection is initiated in the loop 3 after reaching steady condition in the test facility with constant 2.14 l/s (0.19 m/s). Loop 3 is isolated from the primary system by two valves: one upstream the injection position, and another approximately 1 m downstream the hot leg (see Figure 6).

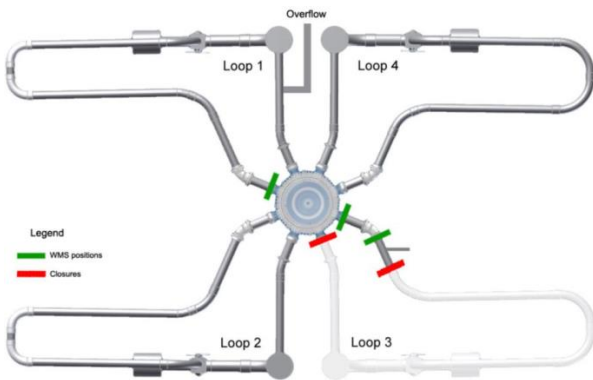


Figure 6. Top view of the ROCOM PKLIIT1.1 experimental setup [2].

4.2 ROCOM modeling with ATHLET-ANSYS CFX

The ATHLET input deck is based on a more generic ROCOM input deck [9]. It represents all four loops of the primary circuit as well as the steam generator simulators and the RPV. The RPV is represented with multiple channels, connected with cross-connection objects. Since in the current work the RPV, the hot legs, and parts of the cold legs are modeled in a detailed way by ANSYS CFX, the ATHLET input deck was simplified. The simplified ATHLET domain consisted of 258 control volumes including the overflow pipe. Figure 7 [10] presents ATHLET (grey) and ANSYS CFX (red) domains in the coupled ATHLET-ANSYS CFX simulation. Both domains were coupled with six coupling

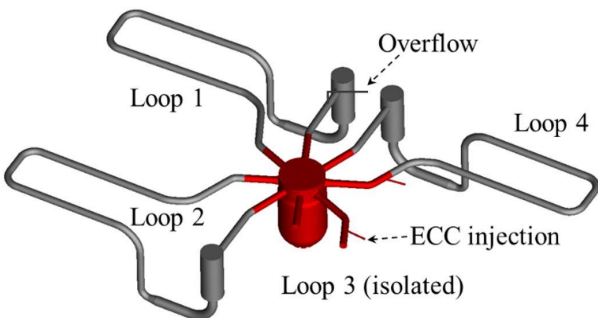


Figure 7. ANSYS CFX (red) and ATHLET (grey) domains in the multiscale model [10].

interfaces located in the three hot and three cold legs. The CFD mesh is based on an existing ROCOM mesh that was improved and extended to 11.3 million elements. In the CFD numerical setup the SST turbulence and full buoyancy models were activated. The selected adaptive time step size for the coupled code run varied within the range 0.001 s – 0.1 s.

4.3 Analysis and validation results for ROCOM PKLIIT Test 1.1

The injected ethanol simultaneously propagates towards the RPV and the cold leg 3 inlet, where an isolation valve is positioned in the facility. The ethanol enters the upper downcomer part and wraps around the inner downcomer wall. The ethanol has a lower density than the water in the primary circuit and is therefore lighter. Large part of it remains in the upper downcomer. A part of the ethanol in the upper downcomer is entrained downwards by the cold water from cold legs 1, 2, and 4, and then transported to the reactor core. The continuous injection increases the ethanol concentration in the ROCOM circuit over the entire injection time, but its distribution in the RPV remains relatively constant: the lighter fluid accumulates in the upper downcomer part, while in the lower downcomer part the heavier water with low ethanol concentration is located. Figure 8 presents the ethanol distribution in ROCOM, calculated with ATHLET-ANSYS CFX

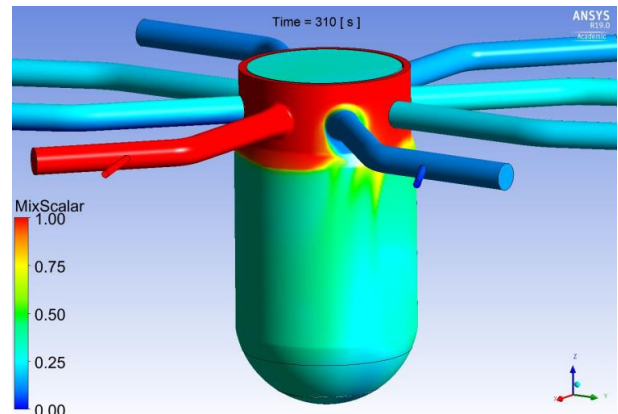


Figure 8. Ethanol distribution in the ROCOM RPV and the attached hot and cold legs at $t = 310$ s.

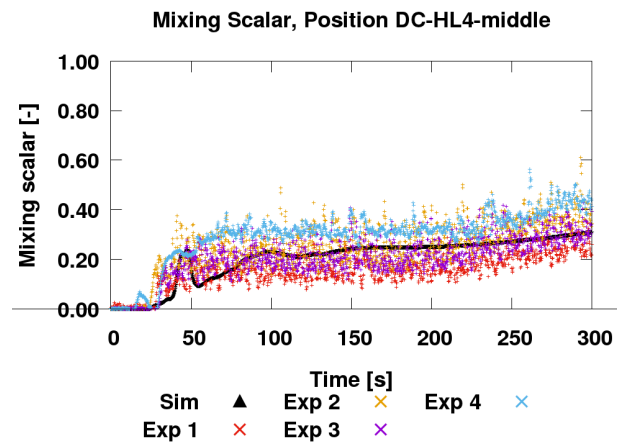


Figure 9. Mixing scalar progression in the ROCOM downcomer under hot leg 4 (1.0 – pure ethanol, 0.0 – pure water).

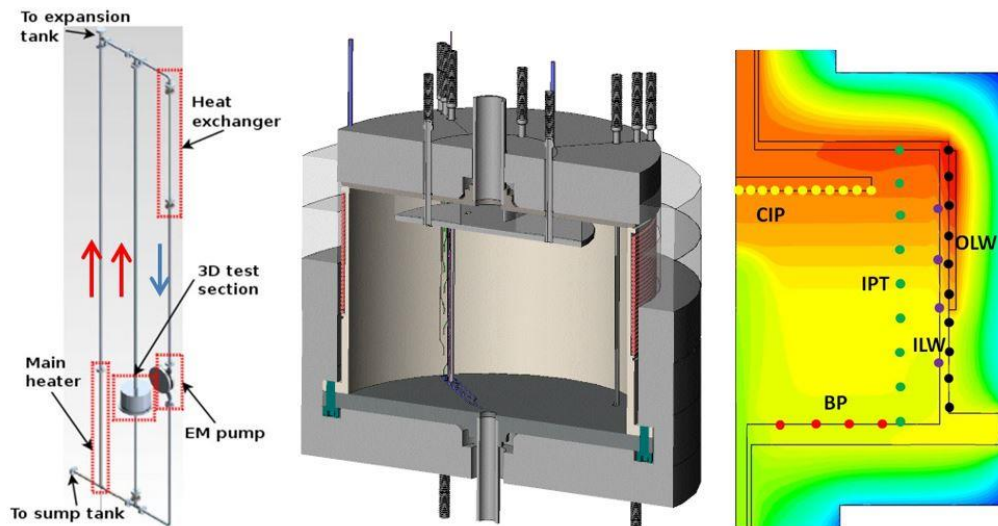


Figure 10. TALL-3D primary circuit with three legs (left), test section (center) and thermocouple positions (right).

at $t = 310$ s. Figure 9 shows the predicted mixing scalar progression along with the four experimental runs with identical boundary conditions (repeatability tests). Good agreement is found ethanol distribution in the system, detailed analysis can be found in [11].

5 Validation of ATHLET-ANSYS CFX on the TALL-3D TG03.S301.04 Experiment

5.1 Description of the TALL-3D facility and the TG03.S301.04 experiment

The TALL-3D thermal hydraulic loop is an integral 7 m high experimental facility, operated by KTH in Stockholm, Sweden. It has two fully instrumented circuits. The primary circuit consists of three vertical legs, filled with lead-bismuth eutectic (LBE) (Figure 10). In the middle leg, a 3D test section is installed, which is the domain of complex 3D flow effects and source for challenging thermal-hydraulic (TH) feedbacks to the rest of the loop. A heater is wrapped around the upper part of the outer test section wall to enhance the buoyancy effects in it at natural circulation conditions. The metal circular inner plate inside the test section pool prevents the inflowing cold LBE from leaving the vessel without extensive mixing with the heated fluid inside it. The test section is positioned in the middle leg, called “test section (TS) leg”. A heated pin is immersed in the lower part of the left primary leg (“main heater (MH) leg”). The heat exchanger (HX) is installed in the upper part of the right leg (“HX leg”). It transfers the heat produced in the primary circuit to the secondary, oil-cooled side. In the TS there are more than 140 TCs, which measure the LBE temperature [3].

The TALL-3D experiment TG03.S301.04 is dedicated to the transition from forced to natural convection in liquid metal reactors. The natural circulation is an important mechanism to remove the generated heat in the primary circuit. In the TG03.S301.04 experiment the main circulation pump is tripped. This leads to a dynamic transient in the facility with local 3D flow phenomena in the TS like LBE mixing and stratification. TG03.S301.04 starts from a

steady state forced convection in the 3D leg with 1.3 kg/s mass flow rate and 235°C at the TS inlet. The TS heater power is kept at constant 4000 W during the whole transient, while the one of the MH is 3200 s.

5.2 TALL-3D modeling with ATHLET-ANSYS CFX

Since the 3D mixing and thermal stratification phenomena occur in the TS, in the coupled simulations it is modeled with ANSYS CFX, and the rest of the facility with ATHLET. A fast-running 2D model is generated, representing a 1° rotational symmetry sector of the TS. The metal plate, the solid walls, the heater, and even the two insulation materials are explicitly modeled with separate structures in ANSYS CFX. The selected mesh for the final simulations has 109,000 elements (Figure 11, right). The ATHLET-ANSYS CFX calculation presented here was performed with the SST turbulence model. The LBE properties are obtained from the OECD “Handbook on Lead-Bismuth Eutectic Alloy and Lead Properties” [12] and implemented in ANSYS CFX. Four priority chains (flow paths) were used for the simulation of the experimental facility with ATHLET. The pressure is controlled with the help of a time dependent volume, which simulates the TALL-3D expansion tank. The secondary circuit is modeled in a simplified way with three pipes. Mass flow rate and enthalpy are specified at the inlet pipe with the help of a FILL component, and the circuit ends with a time dependent volume. The whole facility model comprises approx. 170 control volumes. An adaptive time stepping (0.05-0.1 s) was specified in the coupled calculation.

5.3 Analysis and validation results for the TALL-3D TG03.S301.04 experiment

At $t = 0$ s the electromagnetic pump is tripped with a run-down time of seven seconds, which leads to a rapid mass flow rate decrease in all three primary legs. In the early transient phase, the mass flow rate distribution in both legs is determined not only by the heater powers, but also by the volumes of both components. The MH and the 3D leg compete for the LBE coming from the HX leg. Since the

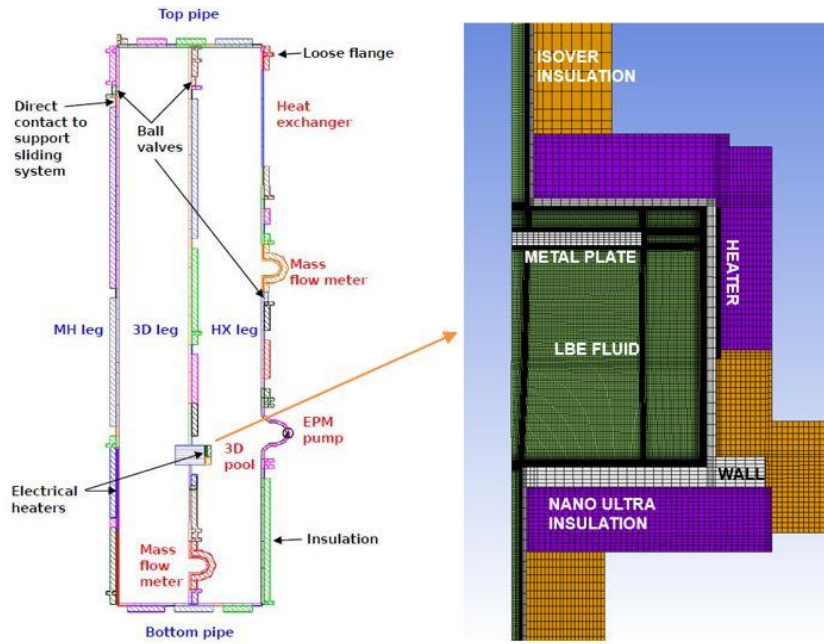


Figure 11. ATHLET-ANSYS CFX model of the TALL-3D facility: ATHLET nodalization (left) and ANSYS CFX mesh (right).

volume of the MH pipe is significantly smaller than the one of the 3D test section pool, the LBE is heated more rapidly in the MH, and its outlet temperature increases faster than the LBE temperature at the outlet of the test section outlet pipe. The correct prediction of the mass flow rate depends strongly on the buoyancy forces, resulting from the density (temperature) distribution, while at the same time the LBE temperatures strongly depend on the mass flow rates. This dependency makes the correct prediction of temperature and mass flow distribution in a multi-leg, buoyancy driven closed system not simple. The resulting temperature peaks in the MH and TS legs are well captured by the coupled code ATHLET-ANSYS CFX. Even more interesting is the temperature progression inside the TS. Figure 12 shows temperature results for the IPT-5 TC, located in the middle of the IPT TC rake (see Figure 11, right). Good agreement between simulation and experimental data was observed for most of the TCs. The thermal stratification that develops inside the TS pool at natural steady state conditions is visible in Figure 13. A well pronounced thermal stratification develops in the TS pool, while the cold

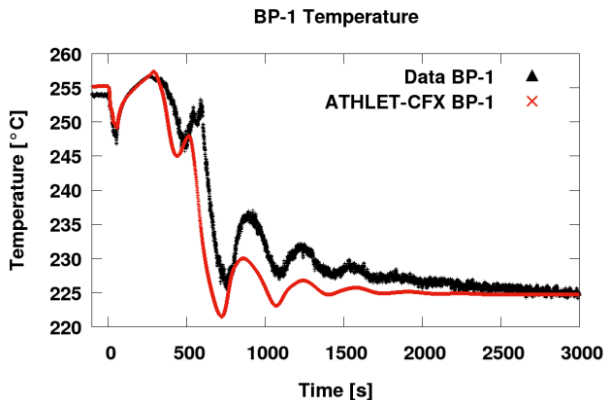


Figure 12. Comparison of the LBE temperature at the IPT-5 location inside the TALL-3D test section.

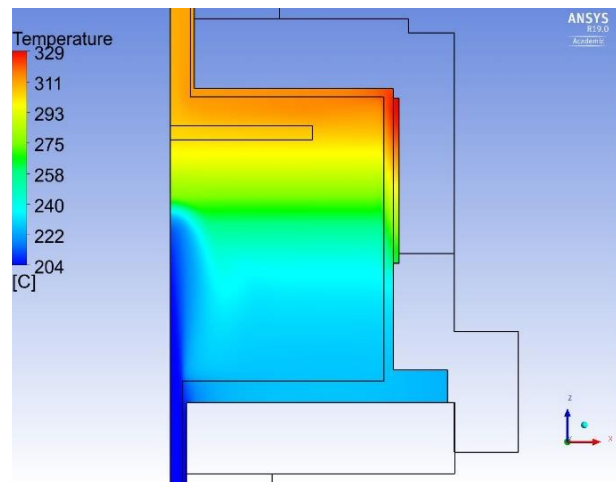


Figure 13. Thermal stratification of the LBE inside the test section at natural circulation conditions ($t = 3000$ s).

LBE jet cannot reach the inner circular plate. More information on this analysis and the advantages of the coupled simulation over the STH one can be found in [13].

6 Multiphysics CFD – CSM Coupling

Under certain conditions, FIV might develop in NPP steam generators, the reactor core, or other components [14, 15]. These phenomena could potentially damage some of the barriers for the safe enclosure of the radioactive inventory, and therefore, need to be carefully analyzed. Such phenomena are evaluated with coupled CFD-CSM codes. In the two-way coupling strategy, both codes exchange data either after or within each time step. The CFD program computes the pressure and temperature acting on the structure surface and provides these to the CSM code. The CSM program solves the equations of motion, considering the boundary conditions provided by CFD, and calculates the structural deformation. The displacement of

each element in x, y, z is passed to the CFD and with the new deformed mesh, the CFD solution for the next time step/coupling iteration is found. Since CFD-CSM simulations might be very expensive concerning CPU time, model order reduction (MOR) techniques are used to provide results at lower computational cost. ANSYS has developed such MOR for the mechanical part and coupled it to the CFD program ANSYS CFX. It is based on the mode-superposition method, which uses the natural frequencies and mode shapes generated from a modal analysis to characterize the dynamic response of a structure to transient or steady harmonic excitations [16].

7 Validation of the Multiphysics Code ANSYS CFX-MOR on Vattenfall Neutron Detection Housing Vibration Experiment

7.1 Description of the Vattenfall neutron detection housing vibration experiment

Within the European VIKING (Vibration ImpaKt in Nuclear power Generation) initiative [17], validation activities for ANSYS CFX-MOR were performed on the Vattenfall BWR Instrumentation Tube Vibration Experiment. The direct practical relevance of this experiment is the vibration of the boiling water reactor BWR in-core neutron detector monitoring system, placed in an instrumentation guide tube (Figure 14). The Vattenfall experimental facility consists of a closed piping loop with a square Plexiglas test section (inner dimensions $80 \text{ mm} \times 80 \text{ mm}$). The latter is built of four scaled BWR bundle walls and an instrumentation guide tube (outer diameter of 8 mm) in the middle. The length of the tube between bottom and top fixations is 1486 mm . The top end of the experimental tube is pinned. At the bottom end the tube is welded into a rigid cross which is then tightened into the steel inlet part of the test section with adjustment screws. Tube displacement, flow rate, and pressure measurements were carried out at Vattenfall [4].

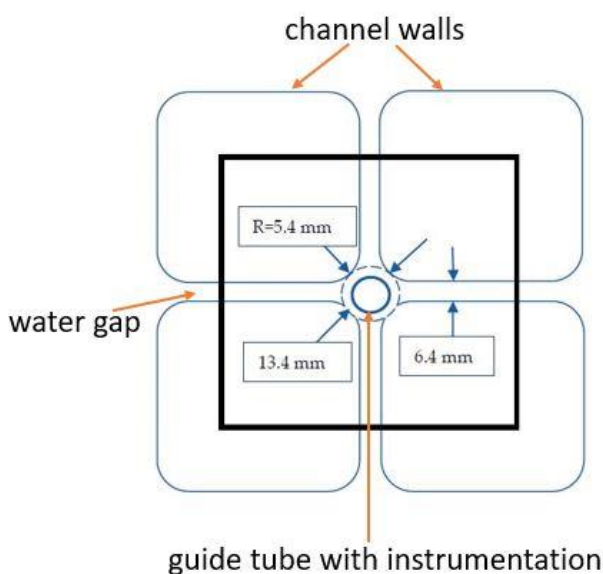


Figure 14. Test section of the Vattenfall neutron detection housing vibration experiment.

7.2 Modeling of the Vattenfall facility with ANSYS CFX-MOR

Despite the test section symmetry, a full FSI model of the Vattenfall experiment was generated, due to the complexity of the tube movement, seen in the experimental data. The modelled domain is inside the black square in Figure 14. The mesh for the FSI simulations consists of 20.8 million elements. To simulate the selected experiment, 5 l/s flow rate and 5% turbulence intensity were specified at the inlet of the test section inlet chamber. Time step size studies showed that reducing the time step size below $2\text{E-}4 \text{ s}$ does not have a noticeable impact on the numerical results, and therefore, this value was used. From the very beginning it was decided to carry out the analysis with Zonal Large Eddy Simulation (ZLES) turbulence model [18]. The advantage of this hybrid turbulence model over the pure LES is the reduced computational effort. At the same time it provides high-resolution in the zone of interest in the computational domain. The structural part of the FSI simulation was performed with the MOR model. A modal analysis with ANSYS Mechanical generated input (eigenfrequencies and mode shapes) in advance for the mechanical MOR.

7.3 Analysis and validation results for the TALL-3D TG03.S301.04 experiment

Figure 15 shows the calculated velocity distribution at $t = 4.4 \text{ s}$, visualized with the help of the Q -criterion. The water flows from the inlet (left) to the outlet chamber (right), which corresponds to the positive z direction. The mean velocity in the inlet chamber is 0.8 m/s . When the water enters the central part of the test section and flows in the gap between the four bundle walls, it is accelerated up to $\sim 6 \text{ m/s}$ due to the significant reduction in the channel cross-section. The instrumentation guide tube starts to vibrate: the predicted RMS value for the tube displacement was 0.063 mm , while the measured one was 0.035 mm . The displacement calculation is considered good as long as the predicted value is within the same order of magnitude of the experimental data. The power spectral density (PSD) showed that the most prominent vibration mode is the first one. It contains most of the vibration energy, and is therefore, the most important for the current analysis. A good

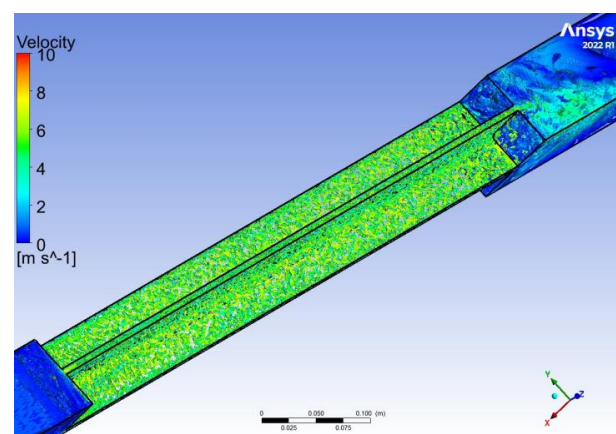


Figure 15. Velocity distribution in the Vattenfall neutron detection housing vibration experiment test section.

agreement between simulation and experiment was observed also here: ANSYS CFX-MOR predicts 11.2 Hz, while the measured value was 10.0 Hz. More information can be found in [19].

8 Validation of the Multiphysics Code ANSYS CFX – Mechanical within the OECD/NEA FSI Benchmark

8.1 Description of the OECD/NEA FSI benchmark

In the proposed by OECD/NEA and OKBM “Afrikantov” FSI Benchmark, cold water (10°C) flows across two in-line cylinders in a narrow rectangular flow channel. The free stream velocity is around 0.5 m/s. As a result of the occurring vortex shedding phenomena, both cylinders start to vibrate. Figure 16 shows a drawing of the FSI test section [5]. The channel is 200 mm high, 30 mm wide and 550 mm long. The cylinders are 198 mm long and have diameters of 7 mm. The distance between both cylinders is greater than 5 diameters. The materials and the instrumentation of the two structures differ a bit, thus leading to slightly different vibration behavior. These are firmly fixed in the bottom plate, while their top ends are not fixed. Cylinders’ accelerations in drag and lift directions, transient fluid velocities, velocity profiles and local pressures were measured in the test section of the OKBM facility.

8.2 Modeling of the OKBM “Afrikantov” FSI facility with ANSYS CFX – Mechanical

In this numerical exercise, the full-order FSI code ANSYS CFX – Mechanical 2021R2 was used. ANSYS CFX and Mechanical are implicitly coupled and perform a number of staggered iterations within each time step. As in the previous case, a two-way coupling scheme is used. In Mechanical, the cylinders were fixed at their bottom ends with the help of *DISPLACEMENT* boundary condition, while the top were let to freely vibrate. The CFD domain comprises the test section channel and part of the inlet chamber. The conical outlet part of the test section was not included in the CFD domain, as it does not have an influence on the flow inside the rectangular channel. The final computational mesh, used for the ANSYS CFX-Mechanical simulations, consisted of 16.7 million elements. Again, the hybrid ZLES turbulence model was employed in this FSI analysis.

8.3 Analysis and validation results for the OECD/NEA FSI benchmark

Zones with different pressure levels form on the cylinders’ surfaces that disturb the local boundary layers. The phenomenon leads to local boundary layer detachment that causes the formation of the vortices. These vortices detach from the cylinder surface and stimulate its vibration. In Figure 17, the vortex shedding from the cylinder surfaces is visualized near the top of the cylinders. The turbulence behind the second cylinder is significantly more complex, because the wake flow from the first cylinder interacts with the vortex formation on the surface of the second cylinder. The predicted RMS acceleration of cylinder 1 is 0.07 m²/s, while the measured one is 0.1 m²/s. The RMS acceleration of cylinder 2 is 0.19 m²/s, the measured one is 0.39 m²/s. The vortex shedding frequency of the first cylinder is calculated to be 17.2 Hz, while the experimental data shows 18.5 Hz. The values for vortex shedding frequency of the second cylinder are very similar. The reduced eigenfrequency of the first cylinder was predicted by the code to be 94.5 Hz, while the measured one was 97.5 Hz. For the second cylinder these are 90.7 Hz and 90.1 Hz. ANSYS CFX-Mechanical manages to capture the vibration behavior of the two in-line cylinders exposed to a cross flow with sufficient accuracy. More comparisons, also in the fluid domain, can be found in [19].

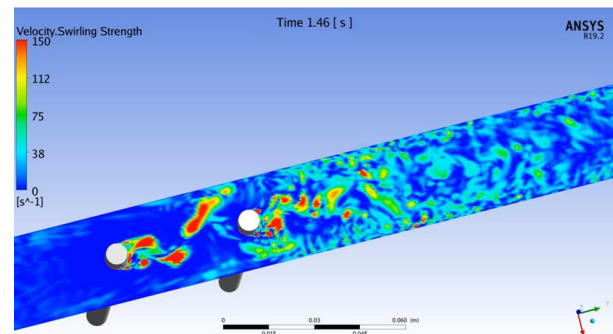


Figure 17. Vortex shedding from the surfaces of both cylinders, visualized with the help of the swirling strength.

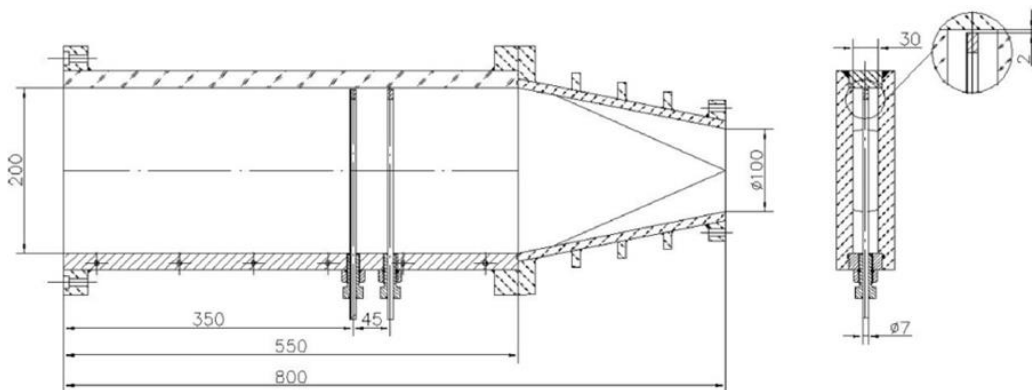


Figure 16. Test section channel of the FSI experimental facility at OKBM “Afrikantov” in Nizhny Novgorod, Russia.

9 Conclusions and Outlook

For many years, GRS has been actively involved in the development, application and validation of STH and CFD stand-alone computer codes for safety assessments. Unfortunately, the application of CFD is still not feasible for the simulation of large nuclear systems and long transients. Therefore, CFD programs are coupled with the GRS' best estimate STH code ATHLET. The multi-scale program ATHLET-ANSYS CFX allows the evaluation of transient NPP behavior with consideration of complex 3D flow and heat transfer phenomena at affordable computational cost. In order to validate the coupled program, large, medium, and small-scale experiments with different coolants were simulated. The presented results for the LSTF, ROCOM and TALL-3D facilities showed good agreement with data and proved the potential of the coupled multiscale program to deal with multidimensional flow phenomena in transient/accident scenarios. The presented work was based on single-phase flow, further work will be mainly focused on the improvement of the two-phase flow coupling and its validation. First attempts in this direction can be found in [20]. The challenges in the two-phase flow are significantly larger not only concerning the numerical stability and the proper choice of two-phase CFD and STH models, but also concerning the representation of a free surface level at the coupling interface, the simulation of evaporation and condensation processes in its vicinity, etc.

For the evaluation of FIV phenomena the multiphysics coupled codes, developed by ANSYS were applied and validated. It turns out that both domains, the fluid and the solid, need to be first represented and then numerically resolved with sufficient detail in order to get accurate predictions. The FSI simulations required meshes of approx. 20 million elements, which automatically led to very high computational times, partly resulting also from the very low time step size. The presented analyses were based on simple slender structures in a single-phase flow, whereas in real life tube and rod bundles with additional support devices such as spacer grids or anti-vibration bars are present in the NPPs. Their influence on the vibration behavior of the rods/tubes is crucial, and therefore, must be taken into account. Hence, more analyses and validation activities on larger geometries are needed to further understand and simulate the complex nature of FIV phenomena in nuclear power reactors. Significant progress in this direction is currently being made in the European GO-VIKING (Gathering expertise On Vibration ImpaKt in Nuclear power Generation) project (2022-2026). The project's main objectives are to expand the expertise in the field of FIV through generation of new experimental data; development, improvement, and validation of FSI methods for FIV evaluation; training of stakeholders in the application of these methods; and synthesizing guidelines for the prediction and assessment of FIV phenomena in nuclear reactors [21].

Acknowledgments

The work related to the liquid metal reactors was partially supported by the European SESAME project (grant agree-

ment No 654935) and projects, funded by the German Federal Ministry of Economic Affairs and Energy.

References

- [1] JAPAN ATOMIC ENERGY AGENCY (2008) Thermal Hydraulic Safety Research Group, Final Data Report of OECD/NEA ROSA Project Test 1-1 (ECCS Water Injection Under Natural Circulation Condition: ST-NC-34 in JAEA). Nuclear Safety Research Centre, JAEA.
- [2] Kliem S., Franz T. (2016) OECD PKL3 Project – Final Report on the ROCOM Tests. HZDR report HZDR\FWO\2016\01 (2016).
- [3] Grishchenko D., Jeltsov M., Kööp K., Karbojian A., Villanueva W., Kudinov P. (2015) The TALL-3D Facility Design and Commissioning Tests for Validation of Coupled STH and CFD Codes. *Nuclear Engineering and Design* **290** 144-153; <https://doi.org/10.1016/j.nucengdes.2014.11.045>.
- [4] Lillberg E., Angele K., Lundqvist G. (2015) Tailored Experiments for Validation of CFD with FSI for Nuclear Applications, Proc. of the NURETH-16 conference, Chicago, USA, August 30 – September 4, 2015.
- [5] Bolshukhin M., Budnikov A., Shmelev E., Kulikov D., Loginov A., Pribaturin N., Lobanov P., Meledin V., Suvorov A., Stulenkov A. (2021) Dynamic Measurements of the Flow and Structure Oscillations to Validate FSI Calculations. *Nuclear Engineering and Design* **381** 111336. <https://doi.org/10.1016/j.nucengdes.2021.111336>.
- [6] Papukchiev A., Lerchl G. (2009) Extension of the Simulation Capabilities of the 1D System Code ATHLET by Coupling with the 3D Software Package ANSYS CFX. In: Proceedings of the NURETH-13 Conference, Kanazawa, Japan, September 27 – October 3, 2009.
- [7] Papukchiev A., Lerchl G. (2010) Development and Implementation of Different Schemes for the Coupling of the System Code ATHLET with the 3D CFD Program ANSYS CFX. In: Proceedings of the NUTHOS-8 Conference, Shanghai, China, October 10-14, 2010.
- [8] Papukchiev A., Lerchl G., Weis J., Scheuerer M., Austregesilo H. (2012) Multiscale Analysis of a Transient Pressurized Thermal Shock Experiment with the Coupled Code ATHLET-ANSYS CFX, *Atw. Internationale Zeitschrift für Kernenergie* **57**(6) 402-409.
- [9] Hristov H. (2012) Numerical Analyses of the ROCOM Tests 1.1 and 1.2 with 1D and 3D ATHLET Models. Technische Notiz, GRS, TN-HRI-01-12.
- [10] Hristov H., Herb J., Papukchiev A. (2019) Analyses of the Flow Mixing Phenomena in a Pressurized Water Reactor by 1D- and 3D Coupled 1D-3D Simulations. In: Proceedings of the NURETH-18 Conference, Portland, USA, August 18-23, 2019.
- [11] Papukchiev A., Yang Z. (2021) Application of the Coupled Code ATHLET-ANSYS CFX for the Simulation of the Flow Mixing Inside the ROCOM Test Facility. *Progress in Nuclear Energy* **137** 103785; <https://doi.org/10.1016/j.pnucene.2021.103785>.
- [12] OECD/NEA (2015) Handbook on Lead-bismuth Eutectic Alloy and Lead Properties, Materials Compatibility, Thermal-hydraulics and Technologies, NEA No. 7268, 2015.
- [13] Papukchiev A., Geffray C., Grishchenko D., Liu C., Kudinov P. (2019) Code Validation Activities within the European SESAME Project and Some Lessons Learned. In: Proceedings of the SESAME Final International Workshop, Petten, The Netherlands, March 19-21, 2019.
- [14] Blevins R.D. (1979) Flow-Induced Vibration in Nuclear Reactors: a Review. *Progress in Nuclear Energy* **4** 25-49.

- [15] Paidoussis M.P. (1980) Flow-Induced Vibrations in Nuclear Reactors, Practical Experience and State of Knowledge. Springer Verlag, Berlin/Heidelberg/New York.
- [16] Einzinger J., Frey Ch., (2014) Bi-directional Fluid-Structure Interaction with Model Order Reduction. Training Course on ANSYS FSI. ANSYS Germany.
- [17] Zwijsen K., Roelofs F., Vivaldi D., Iakovides H., Cioncolini A., Papukchiev A., Benhamadouche S., Deri E., Uribe J., Hadžić H., Lascar C., Chazot B., Goreaud N. (2022) VIKING: A Joint Industry Project on Fluid-Structure Interaction. In: Proceedings of the NURETH-19 conference, online, March 6-11, 2022.
- [18] Menter F. (2015). Best Practice: Scale-Resolving Simulations in ANSYS CFD, Version 2.0. ANSYS.
- [19] Papukchiev A. (2023) GRS Contributions to Flow-Induced Vibrations Related Activities in Europe. *Kerntechnik* **88**(2) 155-173. <https://doi.org/10.1515/kern-2022-0110>.
- [20] Yang Z., Herb J., Papukchiev A. (2023) Two-Phase Coupled 1D-3D ATHLET – OpenFOAM Calculations of the OECD/NEA Large Scale Test Facility ROSA V Test 1.1. In: Proceedings of the NURETH-20 Conference, Washington, USA, 20-25 August 2023.
- [21] Papukchiev A., Zwijsen K., Vivaldi D., Hadžić H., Benhamadouche S., Benguigui W., Planquart P. (2023) The European GO-VIKING Project on Flow-Induced Vibrations. In: Proceedings of the NURETH-20 Conference, Washington, USA, 20-25 August, 2023.