FRONT PAGE

PROJECT FINAL REPORT

Grant Agreement number:	249337
Project acronym:	THINS
Project title:	Thermal-hydraulics of Innovative Nuclear Systems
Funding Scheme:	Collaborating project, "Large-scale integrating project"
Period covered: from	February 01, 2010 **to** January 31, 2015

Name of the scientific representative of the project's co-ordinator[1]**:**

Title:	Prof. Dr. Xu Cheng
Organisation:	Karlsruhe Institute of Technology (KIT)
Tel:	(+49) 721 608 45356
Fax:	(+49) 721 608 45130
E-mail:	xu.cheng@kit.edu

Project website Fehler! Textmarke nicht definiert. **address:** www.ifrt.kit.edu/thins

[1] Usually the contact person of the coordinator as specified in Art. 8.1. of the Grant Agreement.

4.1 Final publishable summary report

4.1.1 Executive summary

For the long-term development of nuclear power, innovative nuclear systems such as Gen-IV reactors and transmutation systems need to be developed for meeting future energy challenges. Thermal-hydraulics is recognized as a key scientific subject in the development of innovative reactor systems. This project is devoted to important crosscutting thermal-hydraulic issues encountered in various innovative nuclear systems, such as advanced reactor core thermal-hydraulics, single phase mixed convection and turbulence, specific multiphase flow, and code coupling and qualification. The main objectives of the project are: (1) Generation of a data base for the development and validation of new models and codes describing the selected crosscutting thermal-hydraulic phenomena. This data base contains both experimental data and data from direct numerical simulations (DNS), (2) Development of new physical models and modelling approaches for more accurate description of the crosscutting thermal-hydraulic phenomena such as heat transfer and flow mixing, turbulent flow modelling for a wide range of Prandtl numbers, and modelling of flows under strong influence of buoyancy, (3) Improvement of the numerical engineering tools and establishment of a numerical platform for the design analysis of the innovative nuclear systems. This platform contains numerical codes of various classes of spatial scales, i.e. system analysis, sub-channel analysis and CFD codes, their coupling and the guidelines for their applications. The project achieved optimum usage of available European resources in experimental facilities, numerical tools and expertise. Through this project a new common platform of research results and research infrastructure, and a synergized infrastructure for thermal-hydraulic research of innovative nuclear systems in Europe is established.

4.1.2 Project context and objectives

The overall objectives of the THINS project are the development and validation of new physical models, improvement and qualification of numerical analysis tools and their application to innovative nuclear systems. To achieve these goals, a thorough review of important thermal-hydraulic phenomena involved in various innovative nuclear systems was carried out, to identify the common thermal-hydraulic challenges that are important to the design of the nuclear systems and need further investigations. Several important, crosscutting thermal-hydraulic topics were identified, i.e. advanced reactor core thermal-hydraulics, single phase mixed convection, single phase turbulence, two-phase flow and code coupling and qualification. The main objectives of the THINS proposal are summarized as below:

- Development of new ***physical models*** for a more accurate description of the thermal-hydraulic phenomena involved in the innovative nuclear systems. Reliability of numerical simulation codes depends mainly on the accuracy of the physical models used. In the frame of this project, more reliable models are developed which are needed in various classes of numerical codes. New correlations for heat transfer and flow mixing in complex fuel assemblies and various types of coolants are be derived for system analysis codes and sub-channel analysis codes. Advanced turbulence models are proposed for a large range of Prandtl numbers. Furthermore, the effects of buoyancy and non-isotropic turbulence are taken into account.
- Development of advanced ***methodology or modelling approaches*** for numerical simulation. The specific features of the innovative nuclear systems require methodology of numerical simulation different from the conventional ones. Liquid metal coolants with low Prandtl number feature the spatial scale for the turbulent energy transport strongly differing from that for the turbulent momentum transport. Accurate simulation of free surface of windowless targets and gas entrainment at liquid pool surface needs more advanced numerical modelling approaches. Coupling of numerical codes of various spatial scales is one important strategy of the future numerical simulation of innovative nuclear systems.
- Establishment of a ***data base*** for the development and validation of new models and codes for a more accurate description of the selected crosscutting thermal-hydraulic phenomena. Generic experiments are

performed in the THINS project to produce a comprehensive data base for the validation purpose. Thus, emphasis is put on advanced measurement techniques, which enable the achievement of both the integral and local test data for the development of macroscopic models such as heat transfer correlations for the system analysis and sub-channel analysis codes, as well as phenomenological models such as turbulence models for CFD codes. In addition, direct numerical simulation (DNS) is performed to provide numerical data base, which is of crucial importance for the development of turbulence models. With this project, a European data base is established for the fundamental thermal-hydraulic issues occurring in the innovative nuclear systems.

- Establishment of an ***experimental platform*** for the thermal-hydraulic research of the innovative nuclear systems. In the frame of the previous European projects, experimental facilities, specifically for the thermal-hydraulics of innovative nuclear systems, have been built and operated at various institutions. The THINS project makes the optimum usage of the available European experimental facilities and expertise, combines the resources available and establishes an European experimental platform. During the THINS project, advanced measurement techniques such as local velocity measurement in liquid metals and high resolution laser measurement techniques for boundary flow conditions, are further developed and integrated into the platform. This complements the results achieved with the VELLA project.
- Establishment of a ***numerical platform*** for the design analysis of the innovative nuclear systems. Numerical codes for nuclear thermal-hydraulics cover various classes of spatial scales, i.e. system analysis based mainly on lumped parameter approach, sub-channel analysis specifically for fuel assembly and reactor core and CFD codes for detailed local flow behavior. The existing codes developed for reactors of Gen-II and Gen-III can't directly be applied to the innovative nuclear systems because of the difference in fluids and flow conditions. With the THINS project, more reliable and validated codes are developed based on advanced physical models and numerical methodology. Coupling of the code solutions at various scales and the qualification, including uncertainty analysis, of the individual codes and the coupled code solutions enlarges the applicability and ensure the simulation reliability of the numerical platform.

The THINS project achieves optimum usage of the available European resources in experimental facilities, numerical tools and expertise and establish a new common platform of research results and research infrastructure. One of the main outcomes of the project is a synergized infrastructure for the thermal-hydraulic research of the innovative nuclear systems in Europe.

The THINS project concentrates on several important, crosscutting thermal-hydraulic issues, i.e. advanced reactor core thermal-hydraulics, single phase mixed convection, single phase turbulence, two-phase flow and code coupling and qualification. Accordingly the THINS project consists of the following 6 technical Work packages:

- WP1: Advanced reactor core thermal-hydraulics
- WP2: Single phase mixed convection
- WP3: Single phase turbulence
- WP4: Multi-phase flow
- WP5: Code coupling and qualification
- WP6: Education and training

The goal of WP1 is to provide validated and verified simulation tools of the coolant flow within the reactor core components for typical situations encountered in innovative reactor concepts. The accompanying experiments with adequate instrumentation are performed to deliver comprehensive test data for the validation of the applied sub-channel and CFD codes.

The WP2 aims at settling an exhaustive and reliable data base for the characterization of the phenomenology in large pool under mixed convection conditions, as well as the development of an appropriate methodology for simulating local phenomena involved with both system codes and CFD codes.

WP3 concentrates on single phase turbulence modelling. It provides reference data for low Prandtl number turbulence model development and experimental reference data for high Prandtl number turbulence model development. Various turbulence models, such as selected algebraic heat flux model and non-linear RANS turbulence models are implemented in various CFD codes, tested and validated. Furthermore, advanced LES turbulence modelling approaches are developed for the simulation of non-unity Prandtl number flows and validated based on data obtained in this WP.

WP4 deals with several specific two phase phenomena, i.e. (a) free surface; (b) HLM/water interaction; and (c) gas/graphite transport. Experimental data are provided for the development of physical models and for the validation of numerical codes. In addition, free surface models, suitable near-interface subgrid scale and subscale turbulence models are developed necessary to simulate sharp fluid-vacuum interfaces for the simulation of free surface flows. New model or new codes (non-commercial) are derived for the prediction of the energy released and for fluid-structure interaction relevant for lead/LBE cooled reactors. In addition, models and extension of available models for the analysis of graphite dust transport, inelastic particle-wall interaction, deposition and resuspension are developed and validated using test data obtained in this work package, to improve CFD analysis of graphite dust behaviour relevant for (V)HTR.

The final results of WP5 are the development and validation of new coupled System-CFD code solutions based on both SFR PHENIX ultimate test and TALL-3D test for reliable prediction of steady state and transient multi-scale thermal-hydraulic phenomena in LM cooled reactor systems. The used system codes are CATHARE, ATHLET and RELAP and the CFD codes are TRIO-U, OpenFOAM and CFX. An improved statistical tool is provided which is designed for the uncertainty analysis of multi-scale coupled codes. This statistical tool is implemented and integrated in computational environments for verification and validation of the PHENIX and TALL-3D tests.

The main targets of WP6 are (a) Preparation of PhD students and other involved students for their tasks within the project, (b) Training of PhD students involved in the project and students attending master courses at partner universities by lectures on thermal hydraulics, computational fluid dynamics and reactor safety, (c) establishment of a scientific platform for young researchers (Master students and PhD students) for exchange of results and for gaining experience in research and in presentation of their research results in front of international audience, and (d) enhancement of results dissemination.

At the end of the project, extensive results have been obtained. Except the E-SCAPE facility, all other 11 test facilities have produced test data related to various cross-cutting phenomena. A large test data base is thus established for model development and code validation. New correlations or models were developed such as heat transfer coefficient, friction factor, turbulence Prandtl number and algebraic heat flux model. They were (partially) validated based on the established test data base and also implemented in numerical codes. In general, the main objectives of the THINS project are well fulfilled.

4.1.3 Main S&T results

The first five WPs are devoted to the individual crosscutting issues. Experiments are foreseen to provide experimental evidence and fundamental test data base. New physical models are developed and further employed to improve codes. The last WP is devoted to the education and training of young nuclear engineers and researchers.

(A) Advanced reactor core thermal-hydraulics

Design of innovative reactor cores requires detailed analysis of the thermal-hydraulics within the fuel assemblies with high resolution inside individual sub-channels. The goal of this work package is to provide validated and verified simulation tools of the coolant flow within the reactor core components for typical states encountered in liquid metal cooled fuel assembly,

The THINS project considers different numerical approaches applicable to the design analysis of innovative reactor cores. The first approach is based on sub-channel analysis codes. Efforts are made to introduce advanced numerical methods, e.g. coarse grid simulation, and to improve the physical models suitable for specific conditions of innovative reactor cores. The purpose of the second approach is to develop CFD simulation tools with more advanced models than the current state-of-the-art RANS modelling and improved accuracy. The complex nature of heat transfer in pebble bed requires the development of new algorithm techniques to describe the core, as well as robust methods capable to treat convective-conductive-radiative heat transfer with fluid flow across complex structure of pebble bed. In the THINS project a method is proposed based on macroscopic modelling of flow phenomena in pebble bed reactors. The macroscopic properties are obtained by LES simulation technique applying to a geometry consisting of spheres in an array with irregular structure.

Two experiments are considered in this WP to provide test data for the validation purpose, as shown in Figures 1 and 2. Test data in fuel assemblies with liquid metal for mixed and natural convection conditions will be produced with the rod bundle test section, see Figure 1, performed in the KALLA laboratory of KIT. Heat transfer tests on single rod with textured surface, see Figure 2, are performed with local temperature measurement in the gas and on the heated rod surface along with LDA to provide experimental data on integral and on the sub-texture scale. The third test data set was obtained in a pebble bed configuration simulating a “fuel compact” in the HTR/VHTR (Hassan et al., 2008). Velocity fields inside the packed bed are measured using Particle Tracking Velocimetry (PTV) combined with the Matching Refractive Index (MRI) methods.

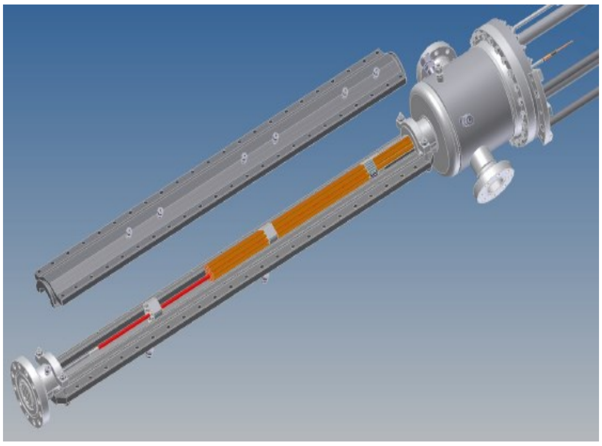


Fig.1: Rod bundle tests with LBE in KIT

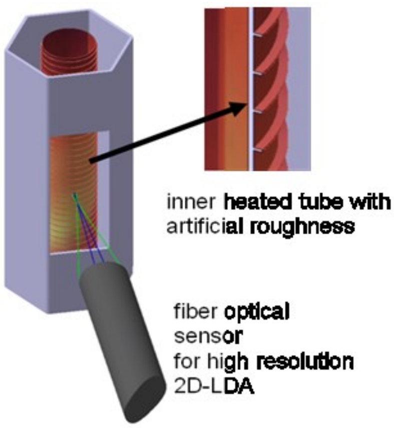


Fig.2: L-STAR tests with air in KIT

(B) Single phase mixed convection

The use of the natural convection to remove residual heat from the core is widely adopted in the Gen-IV reactor concepts that implement passive safety features to enhance reliability. Correct prediction of the onset and stability of the natural convection is essential for the design of the decay heat removal system and for the transient analysis. In addition, the pool-type configuration proposed for most of the innovative Liquid Metal Reactors (LMR) to compact the primary system and to limit the risk of coolant leakage requires numerical tools able to simulate local 3-D phenomena, such as thermal stratification, for the characterization of the thermal loads on the primary vessel and components as well as for the design of in-pool heat exchangers.

In this WP, simulation of dynamic behavior of the innovative nuclear systems is mainly based on system analysis codes originally developed for LWR. The applicability of these codes to support the design process and safety analysis of LMR will be extended and validated. Main extension is focused on the implementation of closure relationships suitable for innovative cooling fluids and for a large spectrum of flow conditions, from forced convection to natural convection.

For the simulation of local 3-D phenomena, CFD approach is applied. The qualification of the CFD models on the separate effect tests and the validation of the system analysis codes on integral tests represent one of the main tasks. To achieve this target, four experiments are planned, one with sodium and the other three with LBE. One of the four test facilities ESCAPE is being constructed in the frame of the THINS project. However, its operation will start after the termination of the THINS project. The other three test facilities will provide test data and are presented in Figures 3 to 5.

The PHENIX end-of-life tests (Gauthe et al. 2012) represent a real reactor configuration and contain complex effects of prototypical systems, see Figure 3. Due to the limitation in measurement instrumentation, however, data available for validation are restricted, e.g. to system dynamic behavior. In the CIRCE (Figure 4) and TALL-3D (Figure 5) test facilities, more sophisticated measurements are applicable. Test data for the qualification for both system codes and 3-D CFD codes are expected. Main focus of at all three experiments is on flows at strong buoyancy effect, i.e. mixed convection conditions.

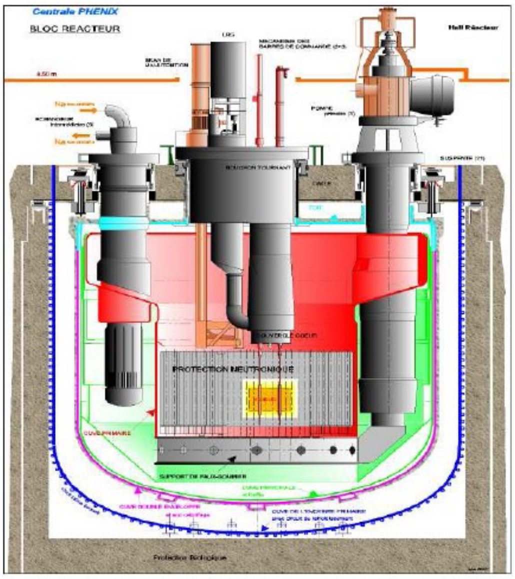


Fig.3: The PHENIX reactor in France

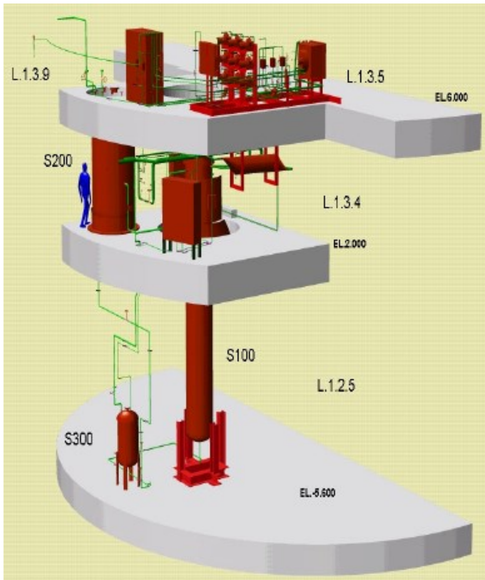


Fig.4: CIRCE facility with LBE in ENEA

Fig.5: TALL facility with LBE in KTH

(C) Single phase turbulence

For the coolants envisaged in the innovative nuclear reactors, usually experiments are very expensive and accurate measurements are challenging or even impossible. Therefore, application of CFD for prediction of various flow characteristics becomes an attractive and complementary practice used in the design and evaluation process of innovative nuclear reactors.

It is well agreed (Roelofs et al. 2014) that one of the key issues ensuring a reliable CFD simulation is the modeling of turbulence. For innovative nuclear systems, two features are important and need to be considered in the turbulence modeling.

(i) The coolants envisaged for advanced reactor systems cover a wide range of fluids with various physical properties, e.g. the molecular Prandtl number varies from the order of 10^{-3} to 10^{3}. This implies the specific behavior and prediction of turbulence and represents a challenging task.

(ii) At normal operating conditions, the fluids at all innovative nuclear systems considered are at single-phase flow conditions.

Therefore, this WP is devoted to single-phase turbulence with the main objective to improve and to develop turbulence models for non-unity Prandtl number flows, their implementation in engineering tools and application to liquid metal and supercritical flows.

Related to innovative nuclear reactors with operation at elevated temperature, this WP focuses on two main issues, i.e. the strong buoyancy effect on turbulence and the occurrence of temperature fluctuation which may lead to thermal fatigue. A large number of CFD codes are applied (see chapter 3). For validation of turbulence models and qualification of CFD codes, three experiments are conducted, as illustrated in Figures 6 to 8.

Both Delight (Figure 6) and SCMix (Figure 7) test facilities uses supercritical (SC) Freon as working fluids. In DeLight facility heat transfer at SC Freon will be investigated with the main emphasis on the investigation of mechanism of heat transfer deterioration. The main purpose of SCMix experiments is the mixing behavior of supercritical fluids under strong density variation and buoyancy effect. The HOMER facility (Figure 8) is designed for study mixing process of two gas components with different densities.

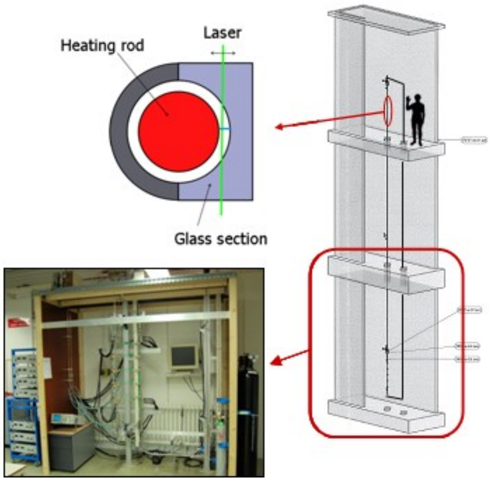


Fig. 6: DeLight facility with SC Freon in DUT

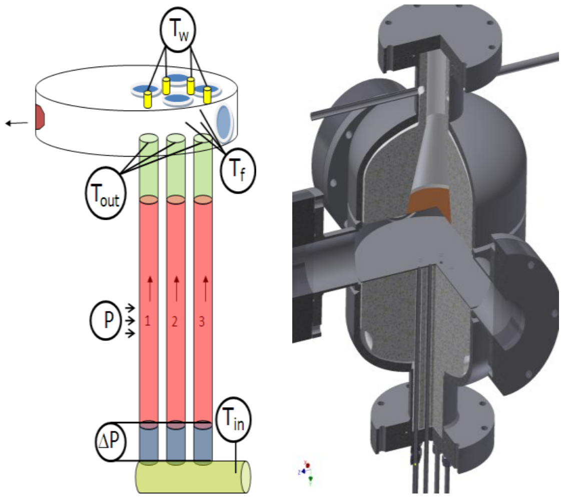


Fig.7: SCMix facility with SC Freon in DUT

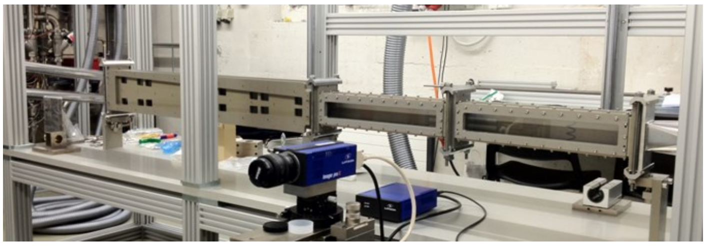


Fig.8: HOMER facility with gas mixture in ETH

(D) Multi-phase flow

Multi-phase flows are encountered in several innovative reactor systems. A sample of these flows forms the basis of the work in this WP. Free surface flows are present in the pool-type liquid metal cooled reactors (LMR). They are also key phenomena in the accelerator driven systems (ADS) with windowless spallation targets. Bubbly flows occur from the possible incidental interaction between water and heavy liquid metal (HLM) in lead alloy cooled reactor systems. In HTR/VHTR, graphite dust is generated by abrasion and transported in the coolant loop.

In this WP advanced free surface modeling methods, such as the ALE-Moving Mesh Algorithm (Maciocco, 2002), will be further developed and validated which are capable of capturing sharp fluid-vacuum interfaces and simulating near-interface subscale turbulence. Concerning the HLM/water interaction, mainly the SIMMER-III code is used to investigate the related phenomena. Complete simulation of the entire HLM/water interaction procedure requires the implementation of additional modules predicting energy release and fluid-structure interaction. For the safety analysis of HTR/VHTR, transportation behavior of contaminated graphite dust generated by abrasion of fuel plays an important role. Development of appropriate CFD models for dust transport simulations is then one of the main tasks of this WP.

To provide test data for model development and numerical codes qualification, three experiments are implemented in this WP, as summarized in Figures 9 to 11. The free surface test at KIT using LBE investigates the characteristics of free surface and the effect of various parameters. The LIFUS facility at ENEA is devoted to study the interaction of LBE with water. It focuses on the measurement of energy release during the LBE/water interaction. At the GPLoop facility at HZDR deposition and resuspension of particles at various flow and surface conditions will be investigated.

Fig.9: Free surface tests with LBE in KIT

Fig.10: LIFUS facility in ENEA

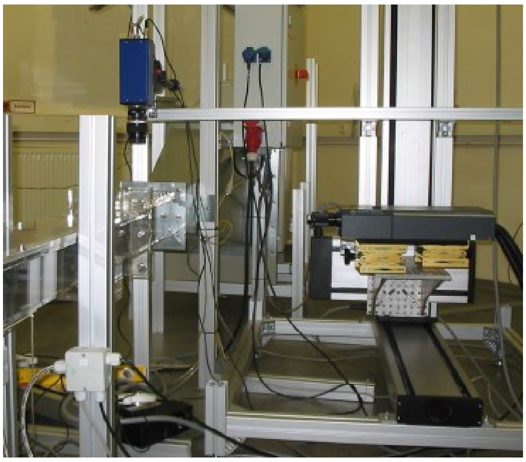


Fig. 11: GPLoop facility for particle deposition and resuspension in HZDR

(E) Code coupling and qualification

The overall objective of the work in this WP is the development and qualification of new multi-scale computational solutions applicable to innovative nuclear systems. A coupled multi-scale solution will combine the merits of various classes of codes and provide design tools with feasible efficiency and high accuracy. This task is devoted to the coupling of system codes with CFD codes and its qualification towards two scenarios entirely dedicated to innovative nuclear systems, i.e. LMR (sodium cooled, lead alloy cooled) with buoyancy effects and transition from forced convection to natural convection.

Two complementary experiments are taken into consideration in the THINS project to provide test data for the qualification of the code coupling solution. The PHENIX reactor scale experiment (Figure 3), with existing ultimate test data on a full scale safety issue scenario, and the HLM TALL-3D (Figure 5) integral scale experiment on a separate effect scale safety issue scenario will produce a unique test data base, which is at the same time complementary to the existing data base for V&V of code coupling solutions.

(F) Education and training

The overall objective of WP6 is the E&T of young researchers and dissemination of the project results. Totally 53 young researchers participated in the project, of which 26 Master students, 20 PhD students and 7 Post-Doc. Four workshops were organized for young researchers to present their results and discuss with each other and also with senior researchers. In addition, training courses were provided for the application of various computer codes and methodology of uncertainty analysis.

4.1.3.1 Technical approach

The methodology used in this project is indicated in Figure 12. Based on phenomenological analysis of the identified crosscutting phenomena, experimental programs were defined with the purpose to provide experimental evidence to a better understanding of physical phenomena and test data for the development of models and validation of codes. Totally 12 experiments have been defined. Due to limited possibility of experiments with low Prandtl number fluids, direct numerical simulation (DNS) will be carried out, to provide additional data for the data base. In parallel, numerical investigations are carried out with three different scales of codes, i.e. system analysis codes, sub-channel analysis codes and CFD codes. Numerical support will be provided to experimental work in various phases, such as selection of measurement techniques, design of test facilities, definition of test matrix and test data analysis.

The data base consisting of both experimental and numerical results can be then applied for the development of physical models and validation of simulation codes. The models considered in this project are for both system analysis codes and CFD codes, e.g. heat transfer and friction pressure drop in liquid metal for system analysis codes, and turbulence models for CFD simulations.

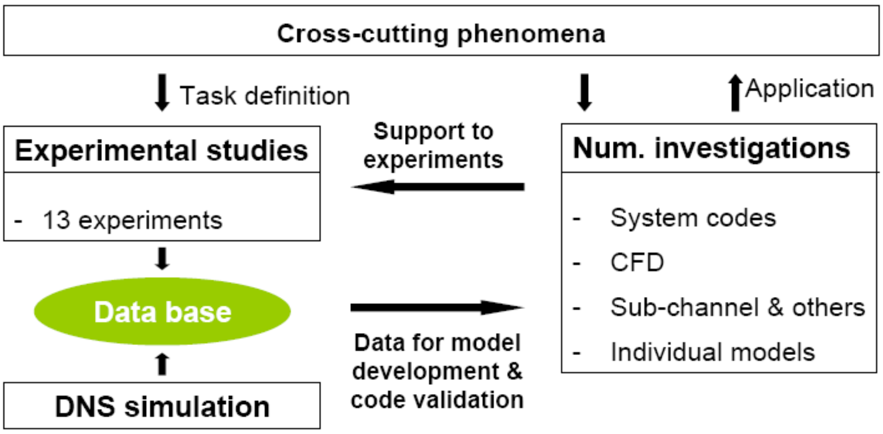


Figure 12: Technical approach applied

To enhance the interaction between the work packages, three ***Clusters*** are introduced in the project management structure, to enhance the interaction between the work packages in which similar or related experimental or numerical approaches may be employed. Therefore, the clusters strongly support the realization of the project objectives. Three clusters have been established, i.e. (i) CFD model and validation; (ii) System code and validation; (iii) Experimental techniques and data bases. The main tasks of clusters are to distribute important information between WPs which might be of general interest for the THINS partners, dealing with experiments, CFD or system code simulations, to organize cluster workshops and to organize required benchmarks.

Clusters

Clusters are introduced in the project management structure, to enhance the interaction between the work packages in which similar or related experimental or numerical approaches are employed. Therefore, the clusters strongly support the realization of the project objectives. Three clusters are established, i.e. (i) CFD model and validation; (ii) System code and validation; (iii) Experimental techniques and data bases. One working group is established for each cluster. The leaders and members of the cluster working groups were determined at the project kick-off meeting. The main tasks of the clusters are:

- To coordinate the interaction between the work packages with respect to experimental and numerical approaches applied;
- To organize benchmarking exercises for the partners of various work packages;
- To arrange the cluster meetings for the information exchange among partners involved in various work packages;
- To ensure the input data and the results transfer between various work packages.

With the efforts of the three clusters, information of all codes used by partners for the THINS projects was collected and evaluated. Some guidelines were proposed. An enhanced exchange in experimental techniques was achieved. Several workshops were organized by clusters for information exchange both among the project partners and with external partners.

4.1.3.2 Experimental activities

Experimental studies are important tools and also necessary boundary conditions to achieve the main target of the project, i.e. development of models and more reliable simulation codes. Twelve experiments have been included in the THINS project and are summarized in Table 1. Real reactor data for sodium as a coolant is provided from the PHENIX end of life test. Six experiments use LBE as fluid. They are dealing with several crosscutting phenomena of LMR, e.g. mixed convection in large pool, mixed heat transfer in rod bundle, HLM/water interaction and free surface flow. It aims at validating system codes, CFD-codes and coupled system/CFD solutions. One experiment is devoted to heat transfer in gas cooled reactor cores with prismatic type fuel structure, whereas another experiment related to gas cooled reactor is dealing with graphite dust behavior, which is of crucial importance for the safety of HTR/VHTR. Two experiments are designed for supercritical water cooled reactors. In the experiment DeLight, heat transfer in single tubes under SCWR conditions will be carried out with the emphasis on heat transfer deterioration and its influencing parameters. The experiment SCMix deals again with the mixing behavior under strong buoyancy effect. At the facility HOMER, buoyancy driven mixing behavior will be investigated using gases of different density.

Related to crosscutting thermal-hydraulic issues it is easily recognized that flow and heat transfer under strong buoyancy effect (mixed convection) is the key topic in this project. It is also important phenomenon in various kinds of innovative nuclear systems, as LMR, GCR and SCWR.

Table 1: Summary of experimental studies

No.	Name of test	Fluids	Main phenomena	Figures
1	PHENIX	Na	System dynamics, pool mixing	3
2	Rod bundle	LBE	Flow & heat transfer behavior	1
3	E-SCAPE	LBE	Pool mixing	--
4	CIRCE	LBE	Pool mixing, heat transfer in bundle	4

5	TALL-3D	LBE	Natural circulation	5
6	Free surface	LBE	Free surface shape	9
7	LIFUS	LBE & water	LBE/H_2O interaction	10
8	L-Star	Air	Local heat transfer	2
9	GPLoop	Air & particles	Particle transportation	11
10	DeLight	SC water	Heat transfer	6
11	SCMix	SC water	Thermal mixing	7
12	HOMER	He & air	Mixing of gases	8

4.1.3.3 Numerical activities

The final goal of the project is the development of advanced physical models, numerical simulation tools and their validation on experimental data. There is strong interaction between numerical activities and experimental studies. As mentioned in Figure 12, numerical activities should in one hand provide support to experimental work and in the other hand utilize the experimental results to improve the numerical tools. In this project a large number of numerical codes are used, which can be divided into three categories, according to their spatial resolution. Table 2 summarizes the numerical codes used.

In the CFD class, eleven codes are selected by the project partners. The focus is clearly on the widely used commercial codes like ANSYS-CFX, Star-CD/Star-CCM+, Fluent, TransAT, and the open accessible code OpenFOAM. However, for certain dedicated purposes also in-house codes like THEMAT, FLUIDITY and SIMMER-III are used. Each experiment is supported by several codes. Different codes might be used by the same partner for different experiments or even for the same experiment, to realize code-to-code comparison purpose.

Five system analysis codes are selected, which are widely applied in the nuclear community. Key focus is clearly on the European codes, i.e. CATHARE from CEA, DYN2B from IRSN and ATHLET from GRS. For analyzing pool type LMR, coupling of system analysis code with CFD code will be realized, i.e. CATHARE with TRIO-U, ATHLET with OpenFOAM, ATHLET with ANSYS-CFX and RELAP with Star-CCM$^+$.

Table 2: Summary of numerical codes used

Codes	Related experiments
CFD	
ANSYS-CFX	TALL-3D, rod bundle, E-SCAPE, L-Star, free surface, LIFUS
Star-CD/Star-CCM$^+$	Rod bundle, TALL-3D, free surface, DeLight

Fluent	E-SCAPE, CIRCE, L-Star, DeLight, GPLoop
OpenFOAM	Rod bundle, free surface, PHENIX, DeLight, HOMER, GPLoop
TransAT	Free surface
SIMMER-III	LIFUS, CIRCE
Armando	Free surface, with ALE-Moving Mesh Algorithm
FLUIDITY	PGLoop, models for particle transportation
System analysis	
RELAP5	CIRCE, E-SCAPE, TALL-3D, also coupling with Star-CCM$^+$
ATHLET	TALL-3D, PHENIX, also coupling with OpenFOM and Ansys-CFX
CATHARE	TALL-3D, PHENIX, also coupling with Trio-U
TRACE	TALL-3D, Rod bundle
DYN2B	PHENIX
Sub-channel analysis	
Coarse mesh CFD	Rod bundle

4.1.3.4 Summary of results achieved

In this paper only a short summary of main results is presented. More details can be found in other publications.

(A) Advanced reactor core thermal hydraulics

In order to improve economic competitiveness and sustainability of innovative reactors new operation conditions are under investigation. These operation conditions are linked to the working cooling fluid, operation temperature, fuel type, and many other challenging issues. Within WP1 the thermal hydraulic behavior within reactor cores utilizing both fuel assemblies and pebble beds are investigated. The investigation aims at providing validated and verified simulation tools of the coolant flow within these reactor core components. Three reference experimental data sets have been generated in addition to available numerical or experimental data to be used for validation of the simulation tools. For fuel assembly investigation, two experiments were considered to provide test data for validation purpose, namely the rod bundle tests with the coolant LBE and L-STAR tests with the coolant air at KIT. In the experimental study employing liquid metal fuel assemblies for mixed and natural convection conditions provide pressure and temperature data, whereas the gas-cooled experiment was focused on heat transfer of a single rod with smooth and textured surfaces. The data include detailed local temperature measurements in the gas and on the heated rod surface. This is complemented with LDA measurements providing excellent experimental data on integral and on the sub-texture scale. A third experimental data set was obtained in a pebble bed configuration simulating a "fuel compact" in the HTR/VHTR (Hassan et al., 2008). Velocity fields inside the packed bed are measured using Particle Tracking Velocimetry (PTV) combined with the Matching Refractive Index (MRI) methods. The tasks of this work package are subdivided according to these experiments, accordingly the subtasks thermal-hydraulics in fuel assemblies is subdivided in two subtasks; heavy liquid metal cooled fuel assemblies and gas cooled fuel assemblies. The thermal-hydraulics in pebble beds is considered in a separate task. Below a summary description of each subtask including context and objectives as well as main S & T results are given.

Heavy liquid metal cooled fuel assemblies

The HLM-cooled rod bundle experiment is setup in the KALLA (Karlsruhe lead laboratory) at KIT for the investigation of both mixed and natural convection. Data are generated in form of pressure losses and temperature values at certain positions. The experimental data are presented in form of tables and correlations for other partners to validate their simulation tools. Various turbulence modeling approaches of HLM heat transfer were applied by the subtask partners. Results of the different models are validated by comparison to the KALLA experimental results. Numerical CFD results were obtained for the low Prandtl number flow employing a turbulent-Prandtl-number look-up table and cover pipe flow, channel flow, gap flow, rod experiment, and rod bundle flow. All cases have shown good agreement for Nusselt numbers. The CFD model validation for the case of isothermal experiments shows that the CFD can be recommended for the prediction of pressure loss across spacer grids with the same level of accuracy as found in the experiment. Four parameter models developed within THINS are also tested and validated. It was shown that the developed κ-ω-κθ -ωθ model shows advantageous features including great stability and easy implementation on boundary conditions in comparison to the other tested four parameter models. A new CFD methodology, namely the Coarse Grid CFD, which considers non-resolved physics by additional volume forces, was further developed and extended by an anisotropic porosity formulation. The code was firstly validated using the available numerical and experimental data and finally applied in simulations of the 127-pin bundle of MYRRHA. The sub-channel code (coarse-MATRA) is also applied and validated. A number of highlights of the conducted work within WP1 related to rod bundles, both experimentally and numerically, can be found in (Pacio et al. 2014).

Gas cooled fuel assemblies

A gas cooled heated rod with a textured surface was experimentally investigated in the L-STAR facility at KIT. Test data, including heat transfer, pressure drop and flow structures on the sub-texture scale measured with LDA have been collected in the range of 3·103< Re <1·105 including different rod surfaces, i.e. smooth surface and ribbed surface (Gomez et al., 2014). The generated data were used to test the adequateness of the available CFD model for the simulation of high velocity and temperature fields by considering different model parameters. The CFD studies were performed using RANS turbulence modelling in ANSYS Fluent and CFX. The test section was fully simulated including the rectifier located at the cold part inlet of the test section. Different turbulence models like Durbin's v2-f model, scale-resolved-simulation (SRS), LES and SAS were tested. Different turbulent Prandtl numbers from 0.9 (default) to 0.6 were considered. Results for different cases of the experiment were compared to numerical results. For cases employing smooth rod surface good agreement between numerical results and experimental data was observed. For roughened surface cases suitable models are recommended based on the validation results. It was found that the choice of a turbulence model affects the results accuracy, where only poor results were achieved with the realizable k-epsilon (two-layer approach for resolving the wall region). Significantly improved results with Durbin's v2-f model were obtained. More details can be found in (Gomez et al. 2014, Suikkanen 2014). Direct simulations with CFX are work in progress to be published soon.

The thermal-hydraulics in pebble beds

Thermal-hydraulics in pebble bed is numerically investigated by using an advanced turbulent modeling technique like mesh-adaptive LES and quasi-direct numerical simulation of a pebble bed configuration. The available pebble bed data by TEES are made available for the project partners for validation of their numerical models. LES and macroscopic models for the flow and heat transfer in pebble beds were considered. A methodology based on LES technique with specialized finite element technique allowing for touching spheres (heavy refinement on edges) was applied. Packing models have been developed for periodic arrays of randomly packed spheres. These packing methods reproduce the well-known packing oscillations in the vicinity of the wall. Also, numerical simulations were performed for two different pebble bed configurations, single cubic pebble bed and a limited sized periodic random pebble bed for which the geometry was provided from the LES study. A series of q-DNS simulations for a single cubic pebble bed were performed as a reference for comparison, where the LES, Hybrid (RANS/LES) and RANS models were compared with the reference q-DNS. The comparison shows that the latter can substantially speed up the simulation compared to LES while still maintaining acceptable accuracy if the correct turbulence modeling approach is selected. Results indicate that a non-isotropic turbulence model is needed for reliable prediction of macroscopic properties. Simulation of heat transfer in wall-bounded flow is difficult due to the unknown heat flux from the bed center to the wall. More detailed results are summarized in (Shams et al. 2013a, Shams et al. 2013b, Pavlidis et al 2013a, Pavlidis et al 2013b).

(B) Single Phase Mixed Convection

The WP2 was aimed at facing the simulation challenges for the single phase mixed convection both at the local scale characterizing the large pool phenomena and at the global scale for the study of the system dynamics. The Sub-Work Package 2.1 was focalized on the qualification of CFD codes and models for the simulation of in-pool phenomena (convection patterns, thermal stratification and fluid-structure thermal exchange). The Sub Work Package 2.2 was devoted to the development and assessment of thermal-hydraulic system codes for the simulation of system dynamics in buoyancy influenced flow conditions by means of code-to-code benchmark and code-to-experiment comparison.

B.1) Mixing and stratification in large pool

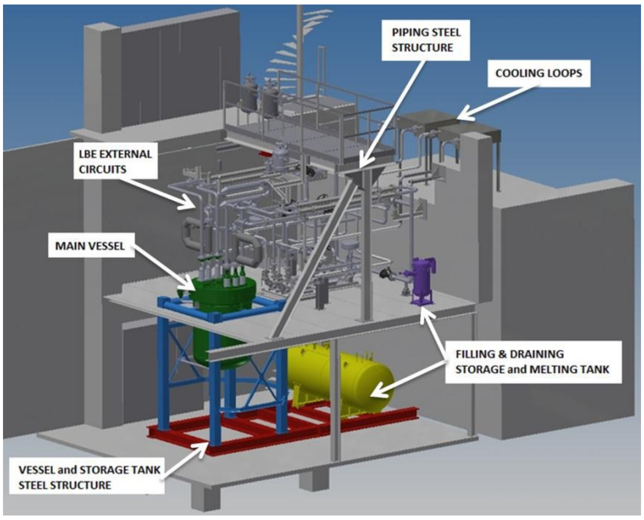
The pool-type configuration proposed for most of the innovative Liquid Metal Reactors (LMR) requires numerical tools able to simulate local 3-D phenomena, such as thermal stratification, for the characterization of thermal loads on the primary vessel and components as well as for the design of in-pool heat exchangers. Furthermore, the correct prediction of the onset and stability of the natural convection in the primary circuit of a pool-type reactor, which might be strongly influenced by mixing and thermal stratification phenomena, is essential for the transient analysis and the design of the decay heat removal system. To this purpose a reliable data base has been established on the basis of experiments (E-SCAPE, CIRCE) with suitable instrumentation. Concerning the numerical activities, CFD modelling strategies for the prediction of the convection patterns and of energy transport in a liquid metal pool have been developed basing on previous studies and literature data. In parallel, a grid-free model has been developed on the basis of Eigen-function expansion techniques for the implementation of pool models in system codes.

SCK-CEN large pool experiment

The E-SCAPE (European SCAled Pool Experiment) facility at SCK•CEN is a thermal hydraulic 1/6 scale model of the MYRRHA reactor, with an electrical core of 100 kW as main power source, cooled by LBE. It plays an important role in the investigation of the feasibility of the passive decay heat removal after reactor shut-down and provides experimental feedback to the designers on the flow patterns in pool-type reactors. Moreover, it enables to benchmark and validate the computational methods for their use with LBE.

The E-SCAPE facility (Fig. 13) has reached an advanced state of construction at the end of the THINS project. All the components and the measurement devices for the vessel and the loops have been purchased and they were available in situ. The mechanical construction of the vessel, including all the internals, the LBE and cooling loops and supporting structure was completed. The placing of the instrumentation is ongoing. In particular, the thermocouples on the vessel, the diaphragm, the baffle and the core (in total around 200 pieces) were already placed and connected. The tracing of the system was completed, the insulation will be placed in March and April 2015. The electrical cabinets were assembled; and the rest of the electrical works will be completed within August 2015. The commissioning will be at the end of August and the facility will be ready for the experiments from September 2015. SCK-CEN will share the outcome of the isothermal tests with the partners of THINS after the project is finalized (as originally agreed).

A set of pre-test analyses based on the as-built models of the facility has been performed using the RELAP5 system code and the Ansys CFX computational fluid dynamic code. The analyses cover the main steady state and transient cases as foreseen for the MYRRHA reactor and some cases specifically distinguishing for E-SCAPE, as the hot plug simulation. The comparison between the pre-test analysis on the E-SCAPE facility and the anticipated behavior in MYRRHA shows that E-SCAPE is capable of simulating the steady state conditions of MYRRHA both in forced circulation and in natural circulation consequent to an event. The transient behavior of E-SCAPE is unavoidably different from the one of MYRRHA, due to the presence of the two external loops and the actuation of the valves to pass from forced to natural circulation. Furthermore, the comparison confirms that RELAP5, despite its 1-D characteristic, can simulate complex 3-D phenomena, however after proper tuning of the models based on the input of the 3-D models. The results of the pre-test analyses will be used later for the comparison with the experimental results. Also the transient behavior will be considered to verify and validate the numerical codes.



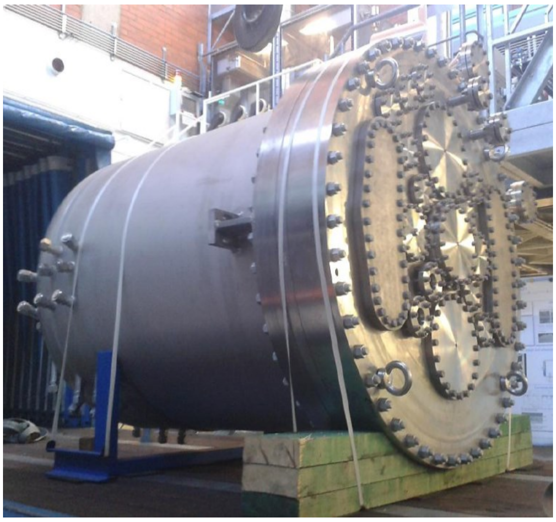


Fig. 13: Isometric view of E-SCAPE facility (left) and main vessel arriving at SCK-CEN (right)

The main purpose of the research activities conducted at HZDR was the development and demonstration of a concept for measuring flow velocities in the E-SCAPE facility by means of the Ultrasonic Doppler Velocimetry (UDV) technique. The flow field in the lower plenum is of specific interest for the investigations. An arrangement of a certain number of sensors is considered to measure velocity profiles along the respective measuring lines. Two components of the fluid velocity can be determined at crossing points of ultrasonic beams. Because the E-SCAPE setup was not available for first test measurements during the life span of the project preparatory measurements were conducted at the LBE test loop METAL:LIC of the

Institute of Physics at the University of Latvia (IPUL). The purpose of this measuring campaign was the demonstration of the concept of sensor installation and use. Owing to the high temperature, the abrasive character of the metal melt and safety reasons the ultrasonic transducer could not be brought into direct contact with the fluid. Therefore, a special socket was used to protect the sensor. The preliminary tests have confirmed the capability of the UDV technique to return the velocity components of the flowing LBE.

CIRCE experiment

In the frame of THINS, ENEA assumed the commitment to design, implement and carry out a large scale integral test campaign on the large scale LBE-cooled CIRCE facility (Fig. 14). This test campaign was aimed at investigating mixed convection and stratification phenomena in safety relevant scenarios such as the transition from nominal flow to natural circulation regime, typical of Decay Heat Removal (DHR) conditions. The CIRCE pool facility has been updated to host a suitable test section in order to reproduce the thermal-hydraulic behavior of a HLM pool-type reactor. The test section basically consists of an electrical bundle (FPS) made up of 37 pins arranged in a hexagonal wrapped lattice. Along the FPS active length, three sections were instrumented to monitor the heat transfer coefficient along the bundle as well as the cladding temperatures at different ranks of the sub-channels. The experimental campaign was designed to study the decay heat removal (DHR) through pool-immersed heat exchangers during accidental events characterized by protected loss of heat sink and loss of flow (PLOHS+LOF) conditions.

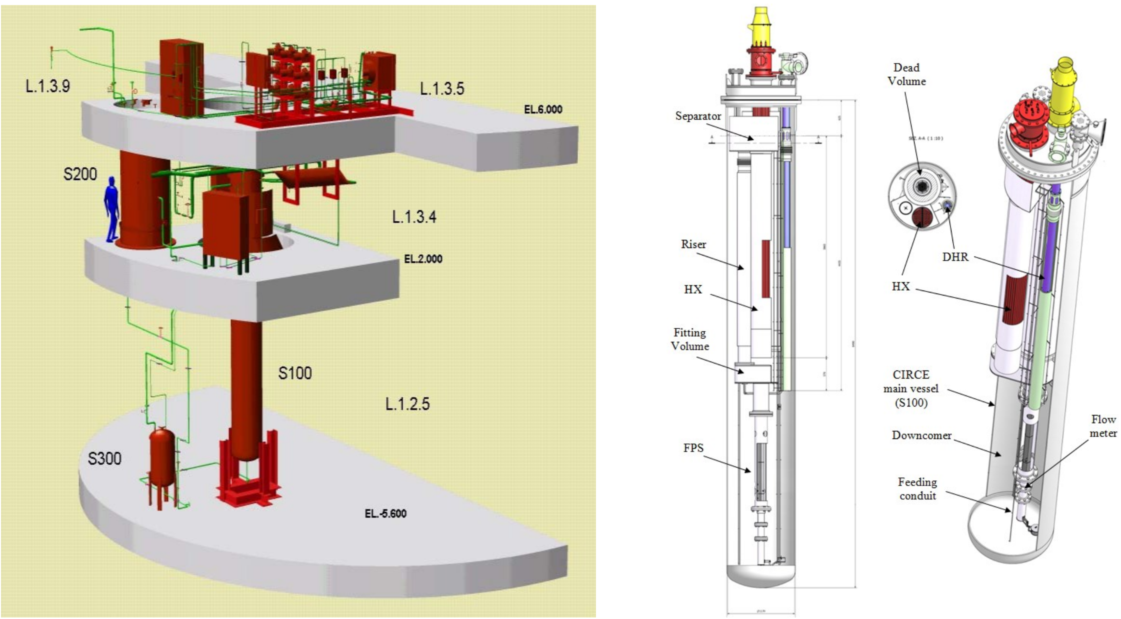


Fig. 14: Isometric view of CIRCE facility (left) and test section for DHR experiments (right)

Experimental data from the most relevant DHR experiments of the campaign, namely Test IV and Test V, were reported and used for both system and CFD code validation purposes in the frame of THINS project by ENEA and the University of Pisa. Temperatures along the three sections of the FPS were reported and the Nusselt number in the FPS sub-channels was investigated together with the pressure losses through the spacer grid. Concerning the main heat exchanger (HX), temperature measurements in the sub-channels were presented, as well as temperatures at the inlet and outlet sections. For the DHR system, temperatures at the entrance and exit section were analyzed both for the primary lead bismuth eutectic (LBE) circuit and for the secondary air side, estimating the thermal power removed by DHR under the formulated accidental scenario. Among the objectives of CIRCE experiment, a great interest was represented by the investigation of in-pool thermal stratification phenomena. For this reason the temperatures in the whole LBE pool were monitored for different elevations and radial locations. The analysis of the experimental data obtained from Tests IV and V put in evidence the occurrence of thermal-hydraulics phenomena already foreseen by pre-test calculations.

<u>Development and assessment of pool mixing and stratification models</u>

System codes usually simplify physics to a one-dimensional description limiting their capabilities when intrinsically three-dimensional phenomena are of importance, such as in case of pool-type thermal-hydraulic systems, where large scale three-dimensional flow pattern exists.

At ENEA, the RELAP5 system code modelling of the pool-type LBE-cooled CIRCE facility has been supported by the analysis performed with the two-dimensional SIMMER-III code and qualified by the post-test analysis of the CIRCE DHR experiments. A mesh-refinement (axial and radial discretization) of the LBE pool volume has been used in the simulation with the RELAP5 system code trying to reproduce mixing and stratification phenomena observed in a pre-test SIMMER-III simulation. The suitability of the one-dimensional RELAP5 and two-dimensional SIMMER-III pool models has been then verified successfully through the post-test analysis of DHR experiments T-IV and T-V performed in CIRCE.

A grid-free pool modelling was proposed and developed at KIT to overcome the limits shown by thermal-hydraulic system codes in the simulation of pool-type systems. The activities were focused on the development of efficient models for the simulation of typical flow scenarios in pool type reactors that subsequently can be included in one-dimensional thermal–hydraulic system codes. The basic modelling strategy consists in reduced order models (ROMs) where the solution of flow variables is approximated by an expansion in terms of global spatial functions. The resulting ROM then consists in a coupled system of ordinary differential equations (ODEs) in time for the coefficients of the expansion functions. The grid-free pool modelling has been tested through a simplified qualitative two-dimensional simulation of a generic LBE-cooled pool-type reactor vessel in transient conditions.

At the University of Pisa the transition from forced to natural circulation condition was extensively investigated together with the thermal stratification phenomena inside the CIRCE pool. Pre-test computational analysis evidenced a well-defined and restricted thermal stratification region between the HX and the DHR exits. The results of the more representative DHR experiments (characterized by different boundary conditions) confirmed the presence in the LBE pool of thermal stratification phenomena, and the region were the thermal gradient occurs was quite well predicted by preliminary simulations. In order to better reproduce several accidental scenarios and improve the accuracy of numerical simulations, a new “two-way” RELAP5-FLUENT coupling tool was developed. The geometry or domain to be analyzed is divided into regions that are modelled using CFD approach and regions that can be reasonably well simulated using a system code. A preliminary application of the developed coupling tool to the Natural Circulation Experiment (NACIE) loop type facility was performed. Obtained results of LBE mass flow rate and pressure differences at inlet and outlet sections of the FPS were compared with RELAP5 stand-alone calculations and data obtained from the experimental campaign performed on the NACIE facility.

B.2) System dynamics

The simulation of the dynamic behavior of the innovative nuclear systems is mainly based on system analysis codes originally developed for LWRs. The applicability of these codes to support the design process and safety analysis of LMRs has been extended and validated in the time frame of THINS project. Main extension was focused on the implementation of closure relationships suitable for innovative cooling fluids and for a large spectrum of flow conditions, from forced convection to natural convection. Validation of system codes has been performed on the basis of already available data (Phenix, TALL, CIRCE) and data provided by new experiments conducted in CIRCE and TALL-3D facilities.

SFR reactors

The ultimate natural circulation transient on the sodium–cooled Phenix reactor was selected for benchmarking and validation of system codes for SFR applications. Participants from three organizations (CEA, IRSN and KIT) calculated this transient using different system codes (CATHARE, DYN2B and ATHLET). To extend the ATHLET code application range to SFRs, a property package calculating sodium thermo-physical and transport properties was implemented into the code by KIT.

Generally speaking, all the system codes predicted reasonably well the different phases of the transient: heating of the lower part of the reactor vessel in the first phase, onset of natural convection and reduction of the whole reactor stratification during the second phase and efficient cooling in the third phase. Nevertheless, all the participants agree that system codes encounter difficulties to predict the evolution of hydraulic paths during the transition between forced convection to natural convection in large pool for sodium reactor with complex 3D geometry. Coupling CFD and system code seems to be a promising way to address such kind of key features in the design and conception of future sodium reactor in the frame of Generation IV. Depending on the meshing, CFD codes could require intensive and expensive CPU resources; therefore the validation of the coupling methodology is an imperative and challenging task. The assessment of the CFD and system codes coupling methodology should remain based on experimental data.

Another important learning of this transient was the challenging task to understand the sensor evolution, especially in large pool and industrial reactor. On forced convection, at nominal power, some strong stratification in particular locations of the reactor vessel (such as at the IHX outlet window or the cold pool) was noticed. After the onset of natural circulation, the field of temperature was strongly modified due to the sodium buoyancy and the complex 3D geometry in the vessel. Moreover, linked to some sensor position and coolant azimuthal influence, it is not always easy to analyze and master the experimental data. Those key learning can be easily extrapolated to other type of reactor, such as HLM reactors with lead or lead-bismuth as coolant.

HLM reactors

Data provided by TALL, TALL-3D and CIRCE facilities were used by KTH and ENEA for the assessment of system codes (RELAP5, TRACE and CATHARE) in HLM reactor applications. The integral experiments considered covers all the possible flow regimes, i.e. forced, mixed and natural circulation conditions in the loop facilities (TALL and TALL-3D) as well as mixing and stratification phenomena in the pool-type facility (CIRCE). The RELAP5 and TRACE system codes were validated on the TALL and TALL-3D experiments by KTH; the CATHARE system code was implemented with lead and LBE properties and validated on the TALL and TALL-3D experiments by ENEA; the RELAP5 system code was assessed on CIRCE experiments by ENEA.

The benchmark of RELAP5 and TRACE on TALL experiments has shown a very good capability to simulate a wide range of transients (start-up, ULOF, ULOH, UTOP, etc.). In particular, both codes predict the initial and final states quite well, even though RELAP5 appears to capture the dynamic phases better than TRACE. The CATHARE simulations of TALL (ULOHS, ULOF and UTOP) gave good results at least in steady-state and long term conditions. The quality of the CATHARE simulations in dynamic conditions is affected by some uncertainties in the experimental facility and experimental data, and occasionally by the lack of information about the instrumentation. For what regards the simulations of TALL-3D carried out by KTH with RELAP5 and ENEA with CATHARE, it must be noted that both codes are one-dimensional and therefore the expectation is that the models have some limitations to match the experimental data, especially when 3D effects inside the test section pool significantly affect the dynamic behavior. Nonetheless, both codes are not provided with oil properties to correctly simulate the secondary side and heat removal from primary loop. The issue is overcome replacing the oil with water in RELAP5 and LBE in CATHARE, and calibrating the heat exchange to consider the different heat capacities of the replaced fluids. The results obtained can be considered good taking into account the above limitations. The RELAP5 heat removal conditions are better calibrated in RELAP5 than in CATHARE model. The initial and final steady states are for most well captured and, during the transients, the qualitative and quantitative behavior of mass flowrates and temperatures are acceptable. A better calibration of the stand-alone models can fairly improve the results while the coupling of system codes with CFD codes can greatly benefit the simulations.

The post-test analysis of DEMETRA and DHR tests conducted on CIRCE facility have demonstrated the suitability of the RELAP5 model to simulate CIRCE experiments under various transient conditions including the transition from forced to natural circulation in the primary circuit. In particular, the stabilized conditions in the primary system are well reproduced while some uncertainties remain in the simulation of transient conditions, likely connected with 3D effects in the LBE pool, namely buoyancy, mixing and temperature stratification, since these phenomena cannot be addressed with the 1D model of RELAP5. The mesh-refinement of the pool for DHR test simulations seems to help the reproducing of the stratification temperature phenomena detected in the test under both forced and natural convection regimes.

(C) Single phase turbulence

For the coolants envisaged in the innovative nuclear reactors, usually experiments are very expensive and accurate measurements are challenging or even impossible. Therefore, application of CFD for prediction of various flow characteristics becomes an attractive and complementary practice used in the design and evaluation process of innovative nuclear reactors. It is well agreed that one of the key issues ensuring a reliable CFD simulation is the modeling of turbulence. For innovative nuclear systems, two features are important and need to be considered in the turbulence modeling:

- The coolants envisaged for advanced reactor systems cover a wide range of fluids with various physical properties, e.g. the molecular Prandtl number varies from the order of 10^{-3} to 10^{3}. This implies the specific behavior and prediction of turbulence and represents a challenging task.
- At normal operating conditions, the fluids in all innovative nuclear systems considered are at single-phase flow conditions.

Therefore, this WP is devoted to single-phase turbulence with the main objective to improve and to develop turbulence models for non-unity Prandtl number flows, their implementation in engineering tools and application to liquid metal and supercritical flows.

Non-unity Prandtl Number Turbulence

For modeling turbulent heat transfer, the current engineering tools apply statistical turbulence closures and adopt the concept of the turbulent Prandtl number based on the Reynolds analogy. Essentially, the turbulent Prandtl number concept is a structural coupling of velocity- and temperature fields, which may be considered as valid only for forced convective flows with Prandtl number of order of unity. In particular cases of liquid metal or supercritical flows, the turbulent Prandtl number concept is not applicable and robust engineering turbulence models are needed. The difficulties mentioned above in applying commercial CFD codes for computation of heat transfer involving liquid metals are well known. The turbulence work package of the THINS project foresees in experiments and direct numerical simulations to support the development and validation of turbulence models for other computational approaches in commonly used engineering CFD tools.

Direct Numerical Simulation (DNS) data from UNIMORE and UCL serve together with experimental data from the DeLight facility at DUT as reference for computational approaches. UCL has focused on DNS for low Prandtl number flow in a rectangular channel. In order to provide reference data at Reynolds numbers as high as possible, they used the fact that when doing LES for momentum, at the same time because of the larger thermal boundary layer, DNS is performed for the thermal field. DNS including conjugate heat transfer for a liquid metal flow has been performed by JSI. In their simulations, they not only model the heat transport in the liquid, but also simulate the heat transport to and in the solid heated wall. UNIMORE on the other hand has performed DNS including separated flow conditions in a wavy wall channel and DNS for a low Reynolds number rod bundle case. Table 3 provides an overview of all the DNS simulations performed as a function of the friction Reynolds number and the Prandtl number.

Table 3: Summary of DNS simulations performed within THINS WP3

Re	Re_τ	Pr=14	Pr = 1	Pr = 0.71	Pr = 0.1	Pr = 0.2	Pr = 0.025	Pr = 0.01
~1500	n.a.				UMR●			
~5500	180		UCL		UCL		UCL	JSI+
~14000	210			UMR~		UMR~	UMR~	
~14000	395							JSI+
~19000	280			UMR~		UMR~	UMR~	
~22000	590		UCL		UCL		UCL UCL*	JSI+ UCL*
~24000	640						UCL*	UCL*
~30000	n.a.	PSI°						
~86000	2000						UCL*	UCL*

UCL : DNS of channel flow by UCL

UCL* : DNS/LES of channel flow by UCL

JSI+ : DNS of channel flow including heat transport in solid wall by JSI

PSI° : LES of pipe flow at supercritical conditions by PSI

UMR~ : DNS of wavy wall channel by UNIMORE

UMR● : DNS of a rod bundle configuration by UNIMORE

Within the experimental 'small loop' supercritical Freon facility of the Technical University of Delft (see figure 15), an LDA technique is used to provide flow field data within the near wall region of a heated rod. The turbulence measurement section has been constructed in a long vertical tube in which supercritical Freon R23 flows vertically upward. The test section consists of an annular flow region around a central heated rod. The aim of the facility is to improve the basic understanding of near-wall heat transfer to supercritical fluids by understanding the turbulence production terms, to (further) develop or select appropriate CFD turbulence models and to validate them. Supporting scoping CFD analyses have been performed by NRG in order to support the design and set-up of the experiment. Although wall temperature measurements are available and have been used by KTH to assess the performance of their newly developed models, the velocity measurements could not be performed within the timeframe of the project. However, results will be provided to the THINS community when they are ready.

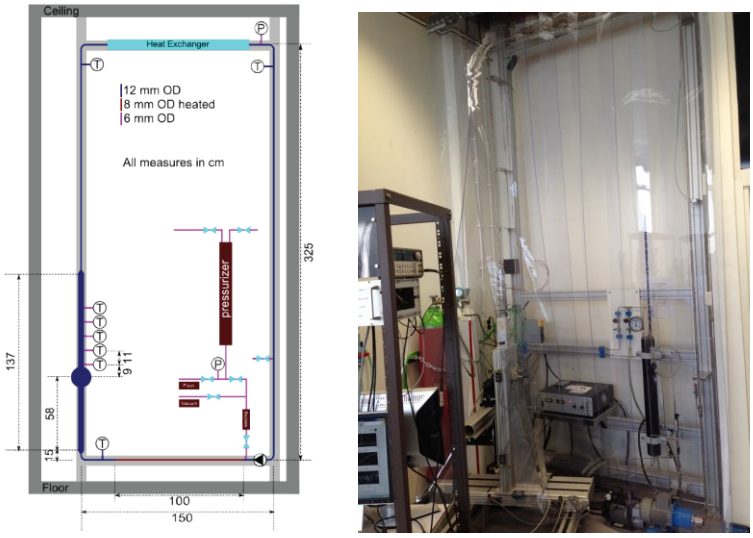


Fig. 15: Layout of the turbulence measurement section in the small loop facility (DUT)

In order to improve the RANS turbulence energy transport models in engineering CFD tools for non-unity Prandtl number fluids, different routes are employed:

- CD-adapco has implemented in their widely spread STAR-CCM+ engineering code a carefully selected, existing advanced Reynolds Averaged Navier-Stokes (RANS) turbulence model based on the model described in 2000 by Kenjeres and Hanjalic. After a first evaluation by NRG and discussion with the model developer, it was decided to implement a modified algebraic heat flux model (AHFM) based on a model described in 2005 by Kenjeres. This AHFM adopts an expression for the turbulent heat flux which is derived from the full differential transport model on the base of a local equilibrium assumption. It retains the fundamental production terms representing the physical mechanisms which generate the turbulent heat flux, therefore permitting to accurately model natural- and mixed-convection flows. The model parameters used in this model have been calibrated for all flow regimes and finally the AHFM-NRG was developed in which one of the model parameters is an expression which depends on the Reynolds and the Prandtl number. This model was validated against available test cases for various Prandtl numbers and different flow regimes. These tests showed good performance, however, it should be mentioned that all of the assessed cases were less complex than reactor scale applications. Therefore, the model will need validation with more test cases and reactor scale applications.
- ASCOMP has implemented various non-linear RANS turbulence models which can be used in combination with the algebraic heat flux model similar to the one developed in 2000 by Hanjalic and Kenjeres in their CFD code TRANSAT. The validation campaign has revealed that the model works fine for unity Prandtl number flow for which it was developed, but experiences problems when applying non-unity Prandtl numbers.
- For the near-wall temperature behavior, UCL derived a new simple wall-function for the temperature at low Pr. Starting from the hypothesis that neither the molecular nor the turbulent diffusivities can be neglected in the near-wall region, and assuming that the turbulent diffusivity is linear, an analytical wall-function can be obtained which transitions smoothly from a linear profile to a logarithmic profile, hence the name mixed law-of-the-wall. Note that the linear turbulent diffusivity hypothesis is

equivalent to assuming a constant turbulent Prandtl number in the near-wall region. Based on the DNS/LES data, a value of 2 was shown to be accurate for low Prandtl numbers in the range 0.01 … 0.025. It was shown that results using the new wall-function and the Kays correlation in the bulk of the channel are as accurate as the wall-resolved results using the same correlation. Since the correlations and the mixed law-of-the-wall where assessed in the channel flow, further validation in more complex geometries should be performed. Also, the mixed law-of-the-wall approach can probably not easily be extended to the mixed and natural convection flow regimes.

To deal with supercritical fluids with Prandtl numbers larger than the order of unity requires development of new RANS turbulence models. To this purpose, first basic understanding is needed of the turbulence production terms. This is obtained from the 'small loop' facility at DUT, supplemented by LES simulations performed at PSI in the in-house PSI-Boil code of PSI. They simulated the experiments of Pis'menny available in open literature for both upward and downward flow. The LES data show good agreement with the experimental data, predicting heat transfer deterioration with good accuracy. These numerical results obviously provide detailed reference data which is very useful for turbulence model development and validation. With the aim of improving the accuracy and/or computational effort in the existing RANS models of supercritical water flows, existing RANS models in different numerical codes were tested by the project partners UniPi, KTH, and NRG. In addition, PSI contributed by implementing and evaluating state-of-the-art and advanced new turbulence models in their in-house THEMAT code. The considerable experience of these project partners was complemented by an extensive literature review of existing experimental data and simulation techniques performed at NRG. This led to the selection of promising innovative turbulence models and CFD approaches. These should enable to improve the accuracy of heat transfer simulations. In summary, it was concluded that the most promising, relatively simple, models are the SST k-ω model and the Lien k-ε model. However, the assessment has also revealed that these models will not provide satisfactory results in all flow regimes without modifications. More promising, but also more complex, identified models include the Myong and Kasagi k-ε model with modifications and the algebraic heat flux model reported by Zhang et al. which was also used by Bae et al. In fact, from the assessment it follows that the latter model is the most promising model. Therefore, this model has been implemented in the in-house THEMAT code at PSI. First tests indeed confirm the promising behaviour of this model. However, in a collaboration between PSI and the university of Pisa it has been shown that also this model will not provide good answers in all flow regimes. Finally, models have been implemented in OpenFOAM which should enable application of CFD techniques to predict heat transfer to supercritical water in complete fuel assemblies at acceptable accuracy. Innovative wall function models have been selected and implemented to achieve this. This should enable improvements either in computational time, in meshing requirements, or in accuracy. One of the models developed by Gerasimov which was implemented showed promising behaviour. However, further improvements are needed. The routes for such improvements have been identified within the project.

Next to all the RANS turbulence model developments, development of LES modeling approaches for simulation of non-unity Prandtl number flows is also being carried out. Based on a two-point correlation method a novel SGS modeling approach for numerical simulations of turbulent flows with non-unity Prandtl numbers is developed by KIT and implemented in OpenFOAM. The new one equation SGS model depends on the local Prandtl number not using the Reynolds analogy between the fields. The implemented model is validated by comparison to DNS data and shows small improvements compared to application of traditional SGS models. Furthermore, a dynamic anisotropic SGS model for the heat flux is implemented by ASCOMP in their commercial TRANSAT code. Validation simulations have shown correct implementation. ASCOMP has also shown the feasibility and robustness of this model in complex practical flows and geometries like T-junctions and rod bundles.

Thermal Fatigue in Innovative Nuclear Systems

Approaches developed for the assessment of thermal fluctuations which might possibly lead to thermal fatigue for current LWRs should be transferred and adapted to innovative reactors. To this purpose, a fundamental experiment dealing with the mixing of different density gases in a rectangular channel, an experiment in a more complex geometry of a small mixing plenum using supercritical fluids, and direct numerical simulations of conjugate heat transfer on temperature fluctuations in liquid metal will be performed to support the development and validation of approaches for innovative nuclear systems.

A fundamental mixing experiment is constructed at PSI similar to their GEMIX (GEneral MIXing) facility. The experimental set-up, called Horizontal Mixing ExpeRiment (HOMER), is shown in figure 16). Experiments have been performed using two separated gas streams at different densities and velocities. PIV (particle image velocimetry) and LIF (laser induced fluorescence) measurement techniques have been used simultaneously to characterize the flow. PSI and NRG have used the experimental data to develop and evaluate in parallel different modeling approaches using RANS and LES techniques. In parallel to this, KIT developed a pseudo-transient LES approach which was validated to available experimental data from literature. This approach allows to increase the computational speed by increasing the computational time step while maintaining a stable solution at the cost of smearing the mixing interface.

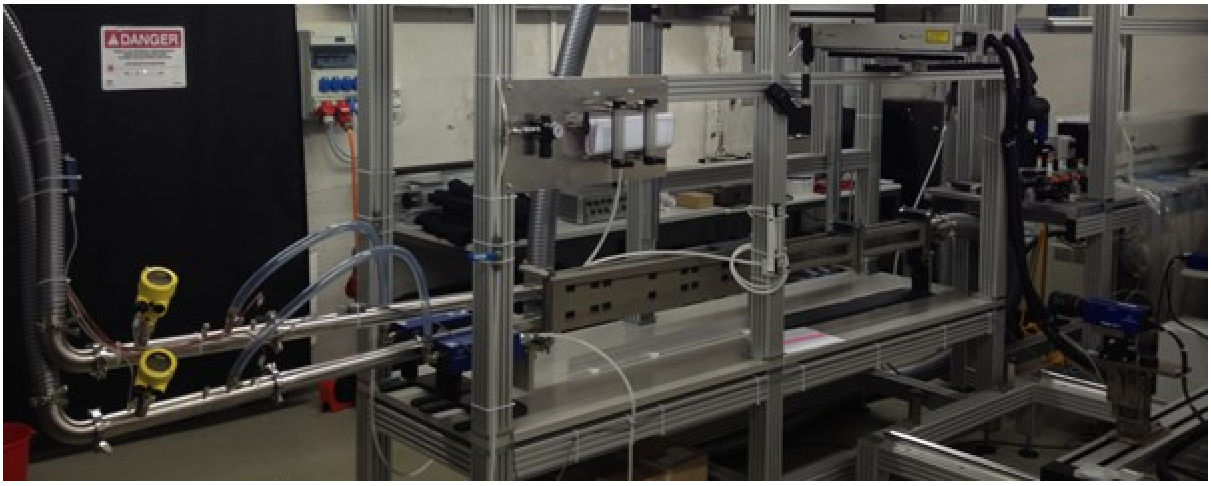
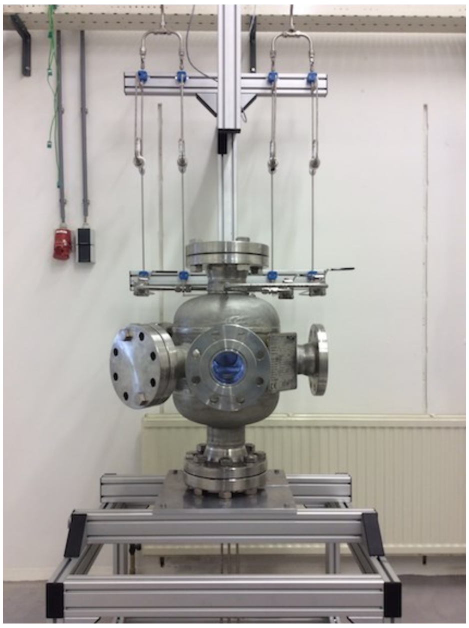


Figure 16 - The HOMER test section (PSI) and a mixing layer simulation (NRG).

Mixing in a core outlet plenum may lead to thermal fatigue damage. Therefore, the development and validation of a numerical modeling approach to simulate the mixing in a core outlet region is of high importance. Within the DeLight facility at the University of Delft, a unique mixing experiment is being set up employing supercritical fluids in which up to three coolant jets may interact in a plenum. Measurements are foreseen using a PIV technique to analyze the turbulent flow field. On top of that, the temperature measurements are foreseen both in the fluid and in the solid structures to analyze thermal fatigue issues.

Due to the high operating pressure, the actual mixing plenum is achieved within a closed pressure vessel with an inner diameter of about 300 mm and a height of about 70 mm. Optical access to the mixing plenum will be created via three sight glasses. Furthermore, a sapphire window at the top of the test section is foreseen to get access to the top wall of the mixing plenum. The conceptual design of the pressure vessel including the mixing plenum is given in figure 17. The design work at the University of Delft is supported by CFD scoping analyses from NRG. First of all, the impingement height and mixing of the jets was characterized in order to fix the main dimensions of the mixing plenum. After that, three dimensional analyses of the complete pressure vessel were performed in order to optimize the flow and mitigate settling of seeding particles which are needed for LDA and/or PIV by design. Finally, NRG has performed pre-test simulations of the final design which should serve the validation purpose once the facility is ready to produce experimental data in the near future.

The influence of conjugate heat transfer on temperature fluctuations in liquid metal is studied by JSI using DNS. In their simulations, they not only model the heat transport in the liquid, but also simulate the heat transport to and in the solid heated wall. Amongst others, they focus on the heat transport between a sodium flow and a stainless steel wall. Their data is used by IRSN to study whether LES performed with relatively coarse meshes not using specific refinements near the wall could provide reasonable results and so reduce the computational effort largely. They conclude that this is indeed possible and a speed up of a factor of 3 can be realised while maintaining satisfactory results. They also show that fluid-wall interactions are very sensitive to boundary conditions and wall properties.



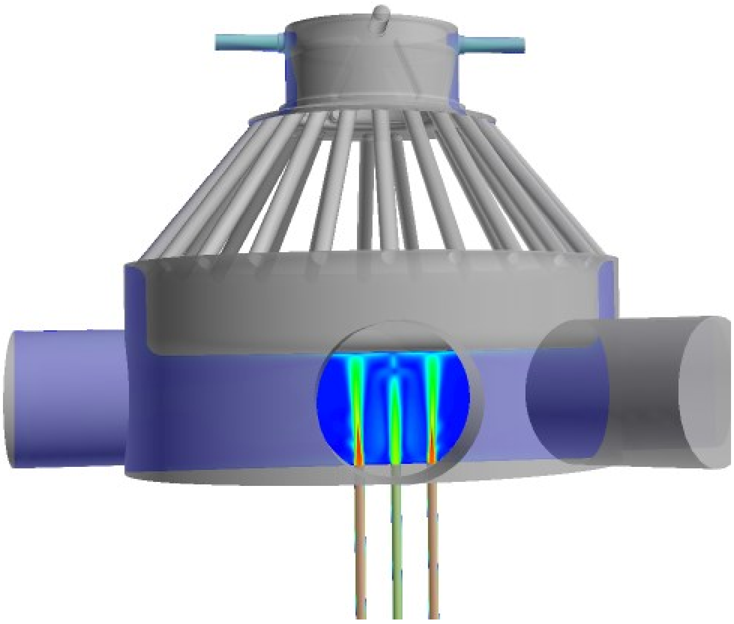


Figure 17 Mixing vessel (DUT) and pre-test simulations (NRG)

(D) Two phase flow

Multiphase flows are encountered in several innovative reactor systems:

- Free-surface flows are present in the pool-type liquid-metal cooled reactors (LMR). They are also key phenomena in the accelerator driven systems (ADS) with windowless spallation targets.
- Bubbly flows occur from the possible incidental interaction between water and heavy liquid metal (HLM) in lead(-ally) cooled reactor systems.
- In HTR/VHTR, graphite dust is generated by abrasion and transported in the coolant loop.

<u>Free-surface flows</u>

Experiments on the highly-turbulent free-surface flow in the funnel-shaped windowless spallation target, as was designed for the MYRRHA ADS, have been performed at KIT at flow rates ranging from 2.0 up to 8.0 l/s. High-speed imaging was used to provide information on surface shape and height of the central recirculation zone. The analysis shows that the outer free surface varies little with the flow rate. The inner free surface shows an inverted conical shape. The cone top height rises linearly with the flow rate and a stable free surface is formed above 7.0 l/s which can be subjected to the proton beam of the MYRRHA reactor.

Concerning the free-surface models developed and used within the project, the Volume-of-Fluid (VOF) cavitation model has proven its ability to predict the free-surface flow with relatively good accuracy and less computational time compared to other models. The development of an improved Arbitrary Lagrangian-Eulerian Moving-Mesh Algorithm (ALE-MMA) for free-surface simulations and its implementation in OpenFOAM has been completed. Computations indicate that the method gives physically reasonable results, although stability issues are still present, mainly connected with the remeshing phase. However, the present stage represents a remarkable step forward. The 3-D Large Eddy and Interface Simulations (LEIS) with the Level-Set method give a rich free-surface image and a good qualitative comparison to the experiments in terms of the interface shape including break-up, however at a high computational cost. Application of the VOF cavitation model with beam heating to the free-surface spallation target geometry indicates that the heat can be successfully removed from the target with moderate surface temperatures. Application of the Smoothed Particle Hydrodynamics code Armando to the ESS target demonstrates suitability of this method for the design and analysis of free-surface flow with fast heating rates.

HLM/Water interaction

The LBE/water interaction that occurs in case of steam generator rupture in an LBE-cooled reactor has been investigated experimentally at injection pressures of 40 and 16 bar in the LIFUS5/Mod2 facility at ENEA that was modified to provide reference data for numerical validation. Next to the injection pressure, the experimental test matrix foresees three quantities for performing single variant tests: temperature of the injected water, cover gas volume and opening time of the injection valve. The measured pressure time trends show a faster pressurization due to colder water injection and lower pressurization in case of higher cover gas volume, in comparison to the reference test.

Pre- and post-test numerical analysis has been carried out by the axisymmetric SIMMER-III code to improve the understanding of the involved phenomena and to confirm the code capabilities in simulating the energy release as a result of the water–LBE interaction. The calculated data showed a qualitatively agreement with the measured values and a faster reaction kinetics due to the modelling assumptions. The key issue to obtain computational pressure time trends in agreement with the experimental data is constituted by a correct estimation of the two-phase pressure drops along the injection line. The temperature time trends numerically simulated for each test show a general cooling anticipation and overestimation on the axis of the reaction vessel. This is due to the impossibility to model with an axisymmetric code the four horizontal cruciform supports that constitute an obstacle against which the water jet impacts and fragments. Moreover, the vapour evolution, expansion and buoyancy in the LBE pool is affected by the code default setting, which could be improved to agree better with the experimental conditions.

Post-test evaluation of selected experimental tests aiming at quantifying accurately the structural effects caused by the interaction between water and the LBE have been also been performed. Two different methodologies have been adopted to accurately reproduce the experimental results: the use of commercial codes for structural mechanics and the development and validation of an in-house code for fluid/structure interaction problems. Good agreement between the numerical results and the experimental data has been achieved. The results demonstrate that for these testing conditions even the simpler approach of a pure structural code is able to properly reproduce the experimental results with adequate accuracy. This is a sign that the interaction between the fluid and the solid is not the driving mechanism that dictates the evolution of the system.

Gas/graphite transport

Single and multilayer particle deposition and resuspension experiments in turbulent channel flows were performed at HZDR. Monodisperse and polydisperse particles were injected into the turbulent flow field and the particle deposition on the channel floor was investigated. Different channel surfaces (smooth and rib-roughened) have been used. The experimental results were used for the development of an Eulerian-Lagrangian particle tracking scheme to compute the particle deposition in turbulent flows. The numerical model was coupled with dry granular mechanics to simulate the growth of a particle multilayer in an obstructed channel flow. Good agreement was found with the experimental results. Furthermore, a numerical model for the computation of multilayer resuspension was developed to simulate the particle detachment. Despite the model simplifications, the numerical results agree well with the experimental data.

Other activities include the assessment of applicability and validity of selected modified Eulerian-Lagrangian RANS CFD approaches for particle transport and deposition. Also, the transportation and deposition of spherical graphite particles in a 90 degree bend pipe is assessed. The particles are tracked using a Lagrangian approach. The results are compared with numerical data from the open literature and the deposition patterns agree largely with the patterns found.

Finally new discretisation methods and mesh-adaptivity methods for multi-fluid flows relevant for particle transport in (V)HTR have been developed and validated. Calculations of the particle transport problem in a channel with period steps have been performed with these new methods.

(E) Code coupling and qualification

The coarse meshes (mostly 0D/1D) used in many system (STH) codes make it difficult to model complex 3D flows. In liquid-metal reactors designs with large liquid plena, such effects can have a large influence on the natural behavior of the reactor during accidental transients such as a loss-of-flow: hence, the safety analysis of these classes of reactors requires the development and qualification of new numerical tools.

As an alternative to the development of new codebases, coupled approaches use existing codes in order to better predict these effects. They take the form of a data exchange between a system code and a CFD code. The latter are suited for modeling regions of interest subject to 3D effects, but lack the capability to describe the rest of the reactor circuit (which remains modeled by the system code). In non-trivial cases, the local phenomena computed by the CFD code affect the global behavior of the system. Hence the need for a two-way data exchange between the codes. In the THINS project, development and validation of coupled approaches was undertaken in the framework of Work Package 5 ("Code Coupling and Qualification"); in addition, uncertainty and sensitivity analyses were performed on the TALL-3D coupled models.

E.1) Validation on the PHENIX natural convection test

The PHENIX pool-type Sodium Fast Reactor (SFR) operated in Marcoule (France) from 1973 to 2009. Before its decommissioning, several end-of-life tests were performed : among them, a Natural Convection Test was undertaken in order to study the transition from forced to natural circulation during a protected loss-of-flow transient. Because this transition involves strong 3D effects in pool-type reactors, it was selected by CEA and KIT to perform reactor-scale validation of their coupled models.

CATHARE / TRIO_U coupled calculation (CEA)

TRIO_U is a CFD code developed in CEA and is especially designed for industrial CFD calculations on structured and non-structured grids. A platform-independent code developed at CEA Grenoble, it is based on an object oriented, intrinsically parallel approach and is coded in C++. For the transient calculation, the hot pool, cold pool and upper-core structure were modeled in CFD (the latter as a solid body). The total number of meshes is 1 230 000 for the three models. Figure 18 (left) shows the meshing of the CFD parts.

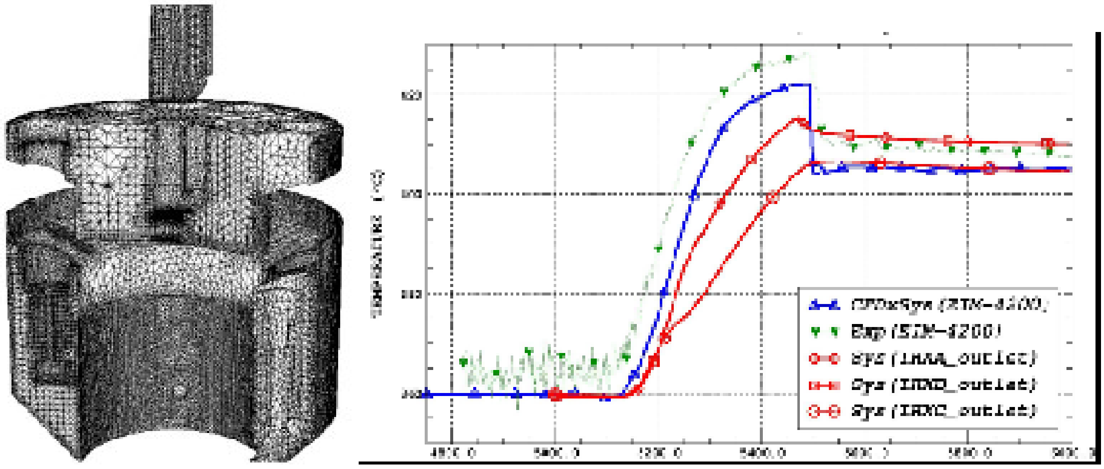


Fig. 18 : CATHARE/Trio_U model : CFD meshes (left), example results (right)

Coupling system and CFD codes enhances numerical tools to capture local phenomena that are inaccessible to system codes. For these local phenomena, complex 3D geometry and buoyancy effects play key roles. Figure 18 (right) gives a good example of such an effect. While the system code doesn't predict the sharp temperature decrease at the IHX primary outlet, the system/CFD coupling is able to reproduce the trend. This better physical prediction comes at the cost of an important increase in the CPU time required to perform the computation (about a week, versus a few hours for the STH-only calculation).

ATHLET / OpenFOAM coupled calculation (KIT)

The coupling methodology uses a domain decomposition approach where neither CFD nor STH have shared domains. This gives the opportunity to handle the CFD and system code quasi standalone. In comparison to this methodology, the domain overlapping method must be mentioned here, where the two regimes are not fully geometrically separated. It is not necessary to run the simulation in synchronized time. Thus, one more degree of freedom and more stability to the individual simulation domain is provided. The two codes provide the full solution (transient case) after the end of each simulation run. In the present simulation the hot plenum is used as CFD regime.

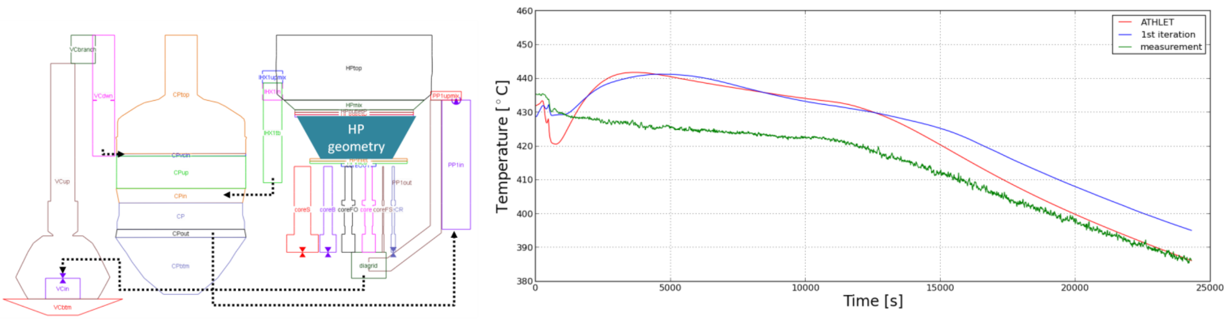


Fig. 19: ATHLET/OpenFOAM: model description (left), example results (right)

The coupled simulation starts with an ATHLET closed circuit stand-alone simulation run (whole transient). Here, the boundary conditions for the later CFD simulation are generated. The CFD regime also simulates the whole transient calculation with the previously generated boundaries. The derived values from the CFD simulation can be transferred to the ATHLET open circuit system where the CFD domain is left out.

The missing values are given from the CFD simulation's results as boundaries. After the ATHLET open circuit simulation of the transient case, new boundaries for the CFD regime are derived and transferred to begin the second CFD simulation. This iteration process continues until the delta of residuals (exchanged values) between the actual iteration step and previous iteration step are a user specified range. In this report one iteration is performed. During verification of the presented coupling methodology, it is observed, one iteration is sufficient for flows without reversals. Otherwise, 2 or 3 iterations (depending on numerical artefacts) are necessary. For the PHENIX natural circulation test, this is not the case for the hot pool region. The effect from CFD regime to system code regime is fully given over the exchanged boundaries. Additional transfer of data is not considered here. The correction from OpenFOAM to ATHLET is effective in every iteration.

As a part of a more extensive validation with measurements, here the measured IHX primary inlet temperature is shown. It almost keeps unchanged until about 490s of the transient. The more than 20°C sharp decrease of the measured core outlet temperature right after the reactor scram just leads to about 4°C delayed decrease of the measured IHX primary inlet temperature. This is due to large thermal inertia in the hot pool and as a consequence of this the very slow transient response speed and strong temperature hysteresis effect. Comparatively, the STH/CFD calculated IHX primary inlet temperature shows good agreement in a short term until approximately 1000s. After scram and pump trip, hot sodium leaving the core is only driven by buoyancy forces. Due to thermal stratification, the new build up temperature field expands to the IHX inlet region. This effect is of strong three dimensional behavior as the hot pool has different temperature zones in horizontal and vertical dimension.

E.2) Validation and uncertainty analysis on the TALL-3D experiment

Experiments selected for the validation of STH/CFD coupled approaches should ideally exhibit strong feedback mechanisms between local 3D effects and global system behavior. In the framework of the THINS project, a new experimental facility, TALL-3D was designed, constructed and operated with this goal in mind. This facility contains two parallel, heated legs: a one-dimensional "main heater" (MH) leg and a "3D" leg containing a cylindrical, heated 3D test section. Both legs are connected to an "heat exchanger" (HX) leg containing a pump and heat removal exchanger.

The reference transient selected for validation in WP5 (T01) consists in a forced-to-natural circulation transition during which both legs are heated. Over the course of the transients, flow oscillations are observed between the two legs : these oscillations are strongly affected by the local behavior of the fluid in the 3D test section.

Coupling of RELAP5 and StarCCM+ for TALL3D experiments (KTH)

The STH code used in this work is RELAP5/MOD3.3-LBE. The TALL-3D model for STH consists of primary and secondary side of the loop connected by a heat structure simulating the heat exchanger.

The CFD model consists of 3D test section and the inlet and outlet pipes. Mesh consists of ~75 000 polyhedral (fluid region) and hexahedral (solid region) elements. To account for wall shear effects, 15 layers of prismatic cells are used near the walls in fluid part. STH steady state simulation provides initial boundary conditions for the CFD code.

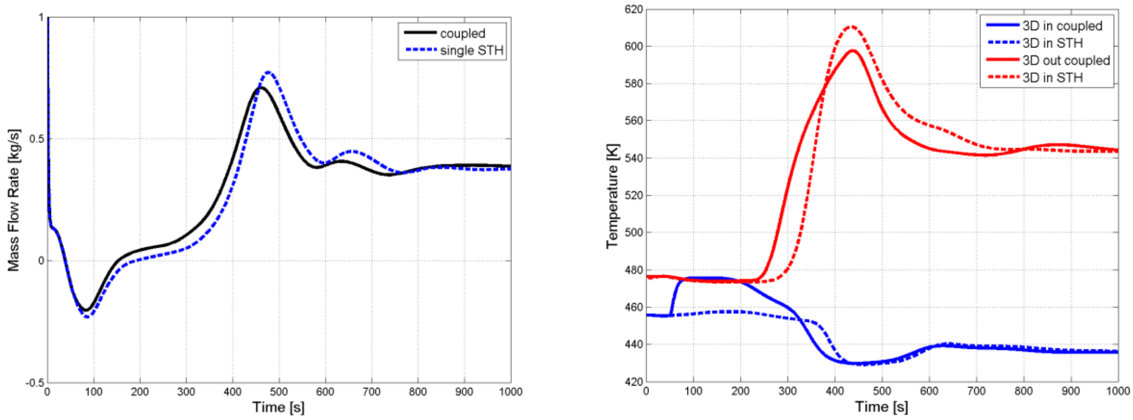


Fig. 20: 3D section flow rate (left) and inlet/outlet temperatures (right) predicted by the STH-only (dotted) and coupled models.

Mass flow rate prediction for T01 is shown in figure 20. As the pump stops, mass flows drop rapidly in each leg. Low flow rate causes the heat-up of the LBE in the 3D pool which, due to flow reversal, is detected also at the bottom inlet. Also, thermal stratification develops at these low flow conditions. As the fluid heats up, the buoyancy forces rise in the 3D leg and flow accelerates again bringing lighter fluid in the vertical pipe which enhances the natural circulation in the whole loop (note that the vertical temperature distribution determines the natural circulation). The 3D leg mass flow rate reaches a peak value around 520 s followed by few oscillations and final steady state.

Coupled simulation results differ from the single STH result and this can be explained by the resolution of the multi-dimensional (i.e. jet vs vortex) flow inside the 3D test section. Hotter fluid at the inlet of the test section is a clear indication of such phenomenon.

Coupled codes enable safety analysts to simulate transients of thousands of seconds' of physical time with CFD accuracy which are helpful in identifying and understanding the complex interactions present in complex flow conditions.

Coupling of ATHLET and ANSYS-CFX for TALL3D experiments (GRS)

Four priority chains (flow paths) were used for the simulation of the whole experimental facility with ATHLET. These describe the primary and secondary circuit of the facility. The 3D test section itself was modeled by 1° hexahedral mesh in ANSYS in order to take advantage of its rotational symmetry: grid sensitivity resulted in the choice of a mesh with 48.000 elements.

For the correct modeling of the buoyant LBE flows in the TALL-3D test section, buoyancy terms in the momentum equation and in the production terms of the turbulence model equations have been included. In the calculations, the SST turbulence model has been used.

With the specified boundary conditions, two calculations have been performed – one with ATHLET and one coupled simulation with ATHLET-ANSYS CFX.

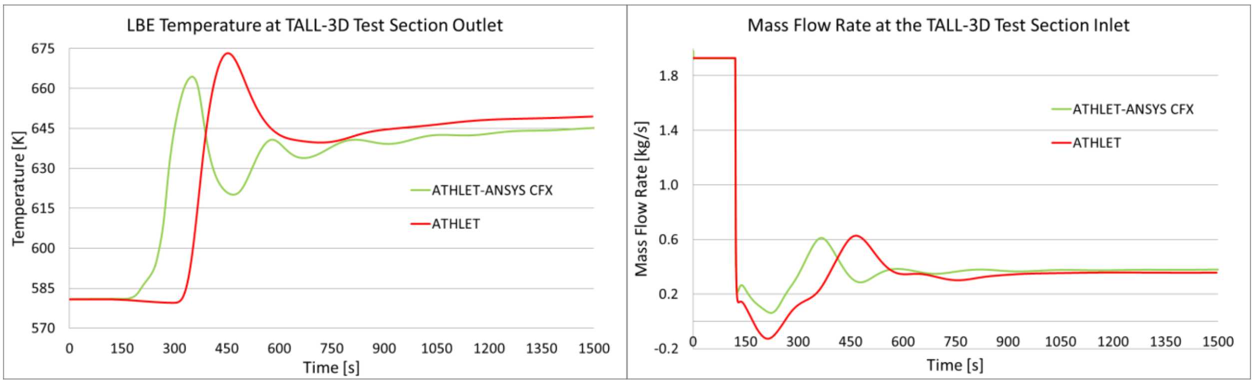


Fig. 21: temperature (left) and mass flow-rate (right) at the outlet of the 3D section during T01

The electromagnetic pump was tripped at 120 s simulation time. Due to the LBE density distribution in the primary circuit and the difference between the geodetic heights of both heaters and the HX, natural LBE circulation occurs in the primary circuit of the TALL-3D loop immediately after the pump trip. In this early phase of the transient, a mass flow increase in the MH leg is observed, while the mass flow rate in the 3D test section leg decreases further (Fig. 21). Because of the intensive natural circulation in the MH leg, less LBE flows in the 3D test section leg. In the stand alone ATHLET calculation, the LBE flow even reverses for approx. 90 s, while in the coupled ATHLET-ANSYS CFX simulation, the LBE flow decreases to 0.06 kg/s, but does not reverse. This imbalance of the LBE flow distribution in the primary circuit occurs, although both heaters are kept operated at 5 kW power each. The reason for this imbalance is related primarily to the different volumes of the MH pipe and the 3D test section pool. Since the volume of the MH pipe is significantly smaller than the one of the 3D test section pool, LBE is heated more rapidly in the MH, and its outlet temperature increases faster than the temperature at the outlet of the test section outlet pipe. The heated LBE is lighter (lower density) and rises faster in the MH pipe. This enhances the natural circulation in this leg, while at the same time it impedes the LBE circulation in the 3D leg.

The temperature at the test section outlet in the ATHLET-ANSYS CFX simulation starts to increase 40 s after pump trip and, in the ATHLET simulation, after 90 s. The reason for the faster temperature increase in the coupled simulation is the positive LBE flow through the 3D test section leg. It is eventually enhanced by more intensive heating in this calculation, since wall structures are not present in the current ANSYS CFX model. This might enhance the natural circulation in the 3D test section leg and eventually hinder the flow reversal in the coupled simulation.

ATHLET and ATHLET-ANSYS CFX results differ from each other, which should be expected, due to the different approaches (1D and 1D-3D), which are implemented. Still the overall prediction of the transient progression in the facility is similar. The experimental data will allow better analysis of the strengths and weaknesses of the different simulation approaches.

E.3) Uncertainty analysis in code coupling

The computer simulation of the behavior of a thermal-hydraulic facility is a challenging task which requires the use of complex physical and mathematical models, and boundary and initial conditions which are affected by uncertainty. This uncertainty, propagated through the calculation process, can significantly impair the accuracy of the results of the simulation. Since the computer model of a facility represents an attempt to simulate its physical behavior as close as possible to reality, the quantification of the uncertainties associated to computer simulations is of key importance for the correct interpretation of the behavior predicted by the simulations.

The analysis of the influence of the uncertainties on the results of a computer simulation of a certain system´s physical behavior can be divided into two parts:

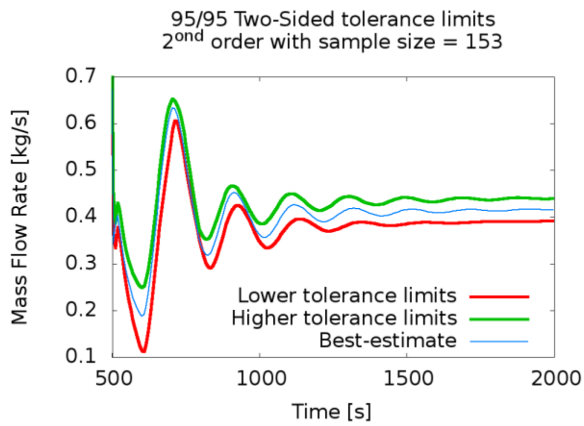
- the Uncertainty Analysis, which investigates the *variability* of the computed results due to the propagation of the uncertainties in models, initial and boundary conditions, etc. along the calculation process.
- the Sensitivity Analysis, which investigates how influential the different models (physical and mathematical), input variables and boundary conditions are on the final results of interest.

Both the Uncertainty and the Sensitivity Analyses provide very useful information about every step of the computer simulation process.

The results of the Uncertainty Analysis statistically quantify the degree of variability affecting the output variables (e.g. temperature at the outlet of a pipe), which is a measure of their uncertainty resulting from the propagation of the modeling, input and boundary uncertainties. When experimental data are available, the comparison of these data with the simulation´s results considering their uncertainties constitute a powerful validation tool. Figure 22 shows the tolerance limits of the mass flow rate in one of the legs of the TALL-3D facility during a transient represented against the best-estimate results. It shows the variability of the output accounted for by the model input uncertainty considered for the TALL-3D facility. Statistically, this limits bound the uncertainty of the mass flow rate with a variability interval [5%,95%] and a confidence of 95%.

The results of the Sensitivity Analysis help identifying the models, inputs, or boundary conditions which most influence the results. Quantification of the influence is done by using statistically based correlation measures. One outcome of the sensitivity analysis is the insight that the most influential variables should be quantified with care since they can account for most of the output´s variability, and their uncertainties will significantly contribute to the output´s uncertainties. Figure 23 shows the influence of different model and input parameters on the maximum temperature observed at a certain location of the TALL-3D facility during a transient. As a measure of sensitivity, the larger the absolute value of the correlation coefficient, the more influential the parameter is.

In the scope of the THINS project, the Technische Universität München (TUM) is in charge of the Uncertainty and Sensitivity Analysis of the TALL-3D experiment. This work is performed using the coupling between ANSYS CFX and ATHLET developed by GRS. The modeling of the TALL-3D facility, cooled by Lead-Bismuth Eutectic, is challenging for the computer codes. Therefore, the work performed by TUM will support the model designers during the whole modeling process and eventually allow the validation of the coupled codes for the thermal hydraulic behavior of liquid metal cooled facility of 4th generation type.



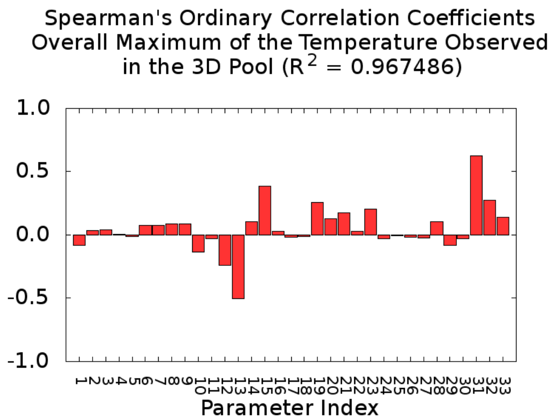


Fig. 22: Uncertainty Analysis *Fig. 23: Sensitivity Analysis*

(F) User's group contribution

A User Group (UG) was established with two members, i.e. AECL from Canada and VKI from Belgium. The main tasks of UG are:

- To use the results of the THINS project to their applications. Such results include new models, simulations tools or experimental data base;
- To contribute to the dissemination and the exploitation of the project results within the involved organizations and beyond;
- To feedback their application experience to the THINS project, so that the THINS project can be dynamically adjusted according to the user requirements, if necessary.

VKI involvement in the THINS FP 7 collaborative project as member of the User Group (UG) is related to WP3 and WP4. Three VKI scientists have actively participated at the different technical meetings. Two young graduate Engineers involved in the VKI Master-After-Master program (academic year 2013-2014) have performed their research master thesis dealing with subjects of the THINS projects. One young graduate Engineer attended a technical meeting. The technical contribution of VKI to the THINS project is summarized as follows:

<u>Contribution to WP 3 – Single phase turbulence:</u>

Temperature wall functions developed by Kader and by Duponcheel was implemented in OPENFOAM. Afterwards the four-equation turbulence model k-□-$k_{□}$-□$_{□}$ for low Prandtl number was developed. The necessity was identified to further fine tune the turbulence thermal diffusivity close to the wall and that an appropriate $k_{□}$-□$_{□}$ boundary condition is required to improve the turbulence model. In addition, a new wall function treatment for the four–equation models was implemented and the results were compared with the literature. The results were presented at two THINS semi-annual technical meetings and will be published in journals.

<u>Contribution to WP 4 – Multi-phase flow:</u>

The VKI has worked on the numerical simulation of water injection into a liquid metal pool, in case of a Steam Generator Tube Rupture (SGTR). VKI had proposed to simulate with the multiphase solver of OpenFOAM the water interaction with liquid metal and to evaluate the performance of CFD with appropriate models for the simulation of the energetic interaction when water is injected in LBE. The proposal was complementary to the work performed in WP 4 (multi-phase model development, implementation and validation) and more specifically task 4.2.2 (HLM/water interaction). The simulations have started with the simulation of the injection of water into HLM without energy released or phase change. Literature has provided experimental results of the water bubble shape and penetration history when injected in liquid metal (Sibamoto experiments at JAEA). The first results have been obtained with ANSYS and have been presented at the 2nd THINS Cluster Workshop of February 2013 at KTH in Stockholm.

Afterwards, the physical phenomena (evaporation, vapor expansion, flashing, splashing,...) responsible for the energetic interaction have been identified and the existing models already available for the prediction of the energy released in the interaction volume during water/LBE interaction have been reviewed (input from THINS in WP 4.2). The complexity of the two-phase flow simulation has led to first the simulation of a single bubble rising a moving liquid, taking into account the phase change. The modelling has been implemented in OPENFOAM. The simulations have been validated using available experimental data and the results have be presented at the technical meeting in Delft (June 2014).

(G) References

Gauthé, P., et al. 2012. The PHENIX Ultimate Natural Convection Test. Proceedings of ICAPP'12, June 2012, Paper 12323

Gómez, R. , et.al., 2014. Experimental results for heated rod in the L-STAR facility. Proc. of THINS 2014 International Workshop, Modena, Italy, January 20-22, 2014, paper-030

Hassan, Y., Dominguez-Ontiveros, E.E., 2008. Flow Visualization in a Pebble Bed Reactor Experiment Using PIV and Refractive Index Matching Techniques. Nuclear Engineering and Design, Vol. 238 (2008). Pp.3080-3085

Pacio, J., et. al. 2014, Heavy-liquid metal heat transfer experiment in a 19-rod bundle with grid spacers, NED paper, Nuclear Engineering and Design 273 (2014) 33–46

Pavlidis, D., Lathouwers, D., 2013a. Realistic packed bed generation using small numbers of spheres, Nuclear Engineering and Design 263, 2013, 172– 178

Pavlidis, D., Lathouwers, D. 2013b. Fluid flow and heat transfer investigation of pebble bed reactors using mesh-adaptive LES Nuclear Engineering and Design 264, 2013, 161– 167

Roelofs F., Shams A., Otic I., Böttcher M., Duponcheel M., Bartosiewicz Y., Lakehal D., Baglietto E., Lardeau S., Cheng X., 2014. Status and Perspective of Turbulent Heat Transfer Modelling for the Industrial Application of Liquid Metal Flows. Proc. of THINS 2014 International Workshop, Modena, Italy, January 20-22, 2014, paper-009.

Shams, A., Roelofs, F., Komen, E.M.J., Baglietto, E., 2013a. Quasi-dirct numerical simulation of a pebble bed configuration. Part I: Flow (velocity) field analysis. Nuclear Enginering and Design, Vol. 263, pp.473-489

Shams, A., Roelofs, F., Komen, E.M.J., Baglietto, E., 2013b. Quasi-dirct numerical simulation of a pebble bed configuration. Part I: Thermal (temperature) field analysis. Nuclear Engineering and Design, Vol. 263, pp.490-499

Shama, A., 2014. An Algebraic Heat Flux Model for Low Prandtl Fluids, Proc. of THINS 2014 International Workshop, Modena, Italy, January 20-22, 2014, paper-002.

Suikkanen, H., 2014. Application and development of numerical methods for the modeling of innovative gas cooled fission reactors, Acta Universitatis Lappeenrantaensis 610, Lappeenranta University of Technology, (2014), ISBN 978-952-265-698-8, ISBN 978-952-265-699-5 (PDF)

4.1.4 The potential impact

The main outcomes of the THINS proposal are the improved numerical tools for engineering applications with more reliable physical models for innovative nuclear systems. The Gen-IV International Forum (GIF) recommended six innovative nuclear energy system concepts for meeting future energy challenges and proposed a Technology Roadmap for Gen-IV nuclear energy systems with the objective to have them available for wide-scale deployment by the year 2030. R&D activities in the first (viability) phase focus on the feasibility of key technologies, and in the second (performance) phase on the ability to make progress toward the desired performance levels. The THINS project fit well in the time schedule of the recommended R&D roadmap of Gen-IV systems.

The Gen-IV R&D roadmap indicates clearly the importance of crosscutting R&D, which must be performed in addition to the system-specific R&D to support the development of a system. The THINS proposal deals with crosscutting thermal-hydraulic issues and completes the existing six crosscutting areas already identified in the Gen-IV roadmap.

The Sustainable Nuclear Energy Technology Platform (SNE-TP) is the European technology platform aimed at promoting the research, development and demonstration of European nuclear fission technologies. The Strategic Research Agenda of SNE-TP has been organized to address the short term (around 2012), the medium term (around 2020) and the long term (2040-2050) R&D challenges and milestones with respect to various systems of fission technologies. It is important to underline that the THINS project has been structured and milestones defined in order to agree with the R&D challenges and milestones indicated in the SRA at a time horizon to 2015. Furthermore, SRA recognizes some crosscutting R&D topics for developing competences and research infrastructures. Some crosscutting thermal-hydraulics issues are classified such as the development of a multi-scale approach for single and two-phase flows; turbulence modelling at local scale and its impact on components' scale. Existing codes can be efficiently adapted to Gen-IV systems as a first practical step towards scoping analysis. In the medium term (2020), efficient sensitivity and uncertainty propagation methods will be developed to handle a larger amount of detailed computational data. General design and analysis tools should be developed for the different Gen-IV systems, taking full benefit from available codes for LWR reactors. In the THINS proposal, new models and codes of various scales, from local scale (CFD) to component scale (system analysis), will be developed based on existing knowledge. Multi-scale code coupling and its V&V are one of the main tasks of the THINS project. This will contribute significantly to the challenges defined in SRA.

4.1.5 Project website

The project website address is: www.ifrt.kit.edu/thins/.

4.1.6 Technical consortium

The technical consortium of the project consists of 24 partners, of those 23 institutions from European Union and one partner from US, as illustrated in Figure 24 and in Table 4. The consortium includes nearly all important institutions in European Union involved in the nuclear thermal-hydraulics, especially dealing with crosscutting topics of thermal-hydraulics of innovative nuclear systems.

Fig. 24: Technical consortium of the THINS project with 24 partners

Table 4: List of project partners

Partner number	**Partner name**	**partner short name**	**Country**
1	Karlsruhe Institute of Technology	KIT	Germany
2	Commissariat à l'Energie Atomique	CEA	France
3	Studiecentrum voor Kernenergie - Centre d'étude de l'Energie Nucléaire	SCK-CEN	Belgium
4	Italian National Agency for New Technologies, Energy and the Environment	ENEA	Italy
5	Nuclear Research and Consultancy Group (NRG)	NRG	Netherlands
6	Paul Scherrer Institute (PSI)	PSI	Switzerland
7	Forschungszentrum Dresden-Rossendorf	FZD	Germany
8	Kungliga Tekniska Högskolan	KTH	Sweden
9	Delft University of Technology	DUT	Netherlands
10	University of Pisa	UniPi	Italy
11	Università degli Studi di Modena e Reggio Emilia (UniMore)	UiMore	Italy
12	University of Bologna	UniBo	Italy
13	Ansaldo Nucleare s.p.a.	Ansaldo	Italy
14	Computational Dynamics Limited	CD-adapco	Great Britain
15	Jozef Stefan Institute	JSI	Slovenia
16	Imperial College of London	ICL	Great Britain
17	ASCOMP GmbH	ASCOMP	Switzerland
18	Institut de Radioprotection et de Sûreté Nucléaire	IRSN	France
19	Gesellschaft für Anlagen- und Reaktorsicherheit mbH	GRS	Germany
20	Center for Advanced Studies, Research and Development in Sardinia	CRS4	Italy
21	Université Catholique de Louvain	UCL	Belgium
22	Technical University Munich	TUM	Germany
23	Lappeenranta University of Technology	LUT	Finland
24	Texas Engineering Experiment Station	TEES	USA