DESIGN AND THE FIRST EXPERIMENTAL DATA FROM SESAME-STAND FOR LEAD SOLIDIFIATION EXPERIMENTS

Tomáš Melichar*,Otakar Frýbort*, Petr Matouš*, Miloslav Bělka*, Vincent Moreau°, Manuela Profir°

*Research Centre Rez, Hlavni 130, Husinec-Rez 250 68, Czech Republic
°CRS4, Scientific and Technological Park, Ed. 1, 09010, Pula, Cagliari, Italy

ABSTRACT

The experimental facility intended for studies of lead natural convection and solidification process in a pool-type geometry has been designed, assembled and commissioned at Research Centre Rez (CVR) within the H2020 SESAME project. The main purpose of the facility is generation of experimental data for validation of computational tools that are being developed for design and safety studies of the lead-cooled Generation IV reactors. The main component of the experimental facility is a cylindrical vessel filled with lead and equipped with electric heaters of power up to 8 kW. An obstacle is placed inside the vessel to ensure relevant natural convection. The heat produced by the heaters is removed through the external surface of the experimental vessel using forced air convection. Apart from detailed measurement of temperature field in the vessel, the facility can also provide data on shape of the frozen structure remained in the experimental vessel by 3D scanning after an experimental run and draining of the liquid fraction. In this paper, the facility will be described in details in order to provide sufficient input needed for preparation of computational models. Lessons learned from the development and commissioning phase of the facility as well as experience with the operation will be also mentioned. In addition, the first experimental data will be presented and compared with data obtained from CFD models developed in parallel at RC Rez using code ANSYS Fluent and at CRS4 using STAR-CCM+. Detected discrepancy between the computational and experimental data and also between the codes will be discussed and resulting recommendations for possible improvements of both the experimental facility and computational models will be indicated.

1 INTRODUCTION

The experimental facility, so-called SESAME-stand, has been designed, assembled and operated at Research Centre Rez (CVR) within the H2020 SESAME project WP3 (Moreau 2019). The main purpose of this facility is generation of experimental data for benchmarking and validation of computational tools that are being developed within the design and safety studies of the liquid metal-cooled nuclear reactors as availablility of such a data is limited. Specifically, this facility works with lead naturally-circulating in a pool-type vessel equipped with electric heaters of power up to 8 kW in the center. The external surface of the experimental vessel (EV) is cooled down by air with controllable mass-flow rate up to 0.5 kg/s, which allows steady-state and transient studies of solidification of the lead under various conditions. Temperature field in the EV is monitored with a large number of thermocouples. Another instrumentation is implemented to measure experimental conditons such as the cooling air parameters or the heaters power and to control auxilliary systems which are intended for filling and draining of the EV. Besides the online measurement of the temperature field, shape of the frozen structure can be also extracted using the 3D scanning after fast draining of liquid content and opening a lid of the EV. Apart from changing of the operational parameters, the facility also allows several options of the geometric layout. An internal obstacle supporting the natural convection in the EV might be re-designed in terms of geometry and replaced. There is also possibility of partial blockage of the air channel which may lead to assymetric cooling conditions. The experimental facility is described below in sufficient detail allowing preparation of a computational model. Experimental data from the first experimental campaign focused on the steady-state measurements will be listed in the form suitable for comparison with computational models. The main challenges and lessons learned from commissioning and experimental phase of the SESAME-stand will be also briefly mentioned.

In parallel with the experimental activity, computational models of the SESAME-stand have been developed using CFD method. A CFD model which is able to deal with natural convection of lead and utilizing an existing solidification module was prepared using code ANSYS Fluent 17 at CVR. The model will be described in the second part of the paper. Results of the CFD model will be compared with the steady-state experimental data. Moreover, results and the main features of a CFD model developed independently at Center for Advanced Studies, Research and Development in Sardinia (CRS4) using code STAR-CCM+ will be also mentioned. Discrepancies between the models will be discussed as well as differences between the models results and the experimental data. Resulting recommendations for possible improvements of both the experimental facility and computational models will be indicated.

2 EXPERIMENTAL FACILITY

In this section, geometric and operational parameters of the SESAME-stand experimental facility will be described in order to provide sufficient input data for preparation of the computational models. In addition, used instrumentation will be also specified and its location will be given for needs of the computational models validation.

2.1 Experimental section

The main part of the facility, where natural convection and freezing of the lead is taking place, is the experimental vessel. Internal diameter of the EV is 300 mm and total height (including the bottom and the lid) is 404 mm. The lead level is approx. 30 mm from the lid (depending on actual temperature level and solidified fraction affecting the lead volume in the EV). Space over the lead volume is filled with inert gas (argon) at a pressure slightly higher than atmospheric to avoid penetration of the air inside the EV and to compensate volumetric changes of the lead. A bottom of the EV has thickness of 25 mm and is equipped with 2° chamfer to facilitate draining. Thickness of the lid is 30 mm and is connected using a flange joint to the top part of the EV which is extended to diameter of 400 mm. External diameter of the EV is 360 mm and the external surface is formed by 48 triangular ribs of height 19.6 mm and inner angle 60° for enhancement of the heat transfer area. The EV is made of cast stainless steel 316, the lid is made of the same material. A replaceable cylindrical obstacle made of stainless steel has height of 180 mm and is placed in the centre of the EV. Internal diameter of the obstacle is 66 mm and external is 97 mm. Four main electric heating rods – main heaters (MH) are located inside the obstacle. Thermal power of each heater is controllable between 0 to 2 kW (total heating power is 8 kW). The heaters are composed of an external steel cover with external diameter of 10 mm and wall thickness of 0.5 mm. A resistive wire made of Kanthal® in the form of helix is generating heat inside the heater. The remaining space in the heater is filled with a ceramic insulator. Total length of a main heater in the EV is 335 mm, the heated length is then 280 mm and the dead end is 5 mm. 19 thermocouple wells with external diameter of 5 mm are distributed all over the EV (see section 2.3). The EV lies on four holders which are connected to a frame of the stand.

The air channel (AC) surrounds the EV. The air flow is driven by a radial fan placed below the EV. The maximum air mass-flow rate is 0.5 kg/s. External diameter of the air channel in the vertical position corresponding to the EV is 450 mm. A conical cover protecting cables coming out of the EV is located in the AC under the EV. Another purpose of the cone is to ensure better air flow distribution. The upper part of the AC acts as a chimney providing ventilation of the hot air from the stand to the experimental hall. This part is equipped with a mass-flow meter (see section 2.3). Partial blockage of the AC might be implemented to ensure asymmetric cooling conditions.

2.2 Auxiliary systems

The experimental facility has several auxiliary systems, which are not active during the experimental campaigns and therefore may not be included in the numerical models, nevertheless they are essential for the operation of the facility. These systems are primarily intended for filling and draining of the EV with lead. The filling/draining section is composed of a storage vessel (SV) and a filling pipeline. The SV is located next to the fan and contains solid lead before experimental operation. The SV is connected with the bottom of the EV using a filling pipeline. All the filling section has to be heated over the lead melting point to avoid undesirable freezing and, therefore, several auxiliary heaters were implemented. The SV is equipped with heating belts with total heating power of 6 kW on the external surface. A heating cord is wrapped around the filling pipeline. Three additional heating cartridges are inserted in holes in the bottom of the EV for the same reason. The empty EV cannot be heated using the MH due to limited heat removal (the MH might be turned on once they are in full contact with the lead). The filling pipeline also acts as a shut-off valve. When the lead in the EV reaches required level, the filling pipeline is sharply cooled down to freeze its content and to hold liquid lead in the EV without need of keeping constant pressure in both vessels.

A gas system is intended to control a pressure in both the EV and SV. The SV pressure control is necessary for filling of the EV. When all the lead in the SV is liquid, pressure over the lead level can be increased in order to push the lead up to the EV. Pressure in the EV is controlled also during normal operation at value slightly higher than the atmospheric to avoid air penetration. The gas system is connected to a vacuum pump for removal of air from the system before melting to avoid lead oxidation. The gas inlet can be reconnected from pure argon to argon/hydrogen mixture for purification reasons. Pushing of the mixture in the system with liquid lead for few hours helps to reduce amount of oxides. Resulting vapours are then sucked out of the system. This procedure is carried out before each experimental run.

2.3 Instrumentation

Totally 21 thermocouple (TC) probes are inserted inside the EV. These sensors are intended for monitoring of temperature field of the lead during the experiment and are therefore the most important for benchmarking purposes. Distribution of the probes is shown in Figure 1. Probes 1-12 are attached to the EV bottom, probes 13-21 to the lid. Each probe has 5 measuring points in vertical direction, only probe 21, which is located in the centre of the EV, has 9 measuring points. Pitch between the measuring points is 20 mm, the first measuring point is 5 mm from the top of a probe. All TCs are plugged in a data acquisition system (DAQ) based on National Instruments PXI platform. Is consists of a chassis, controller and PXIe cards for the thermocouples and the other analog signals.

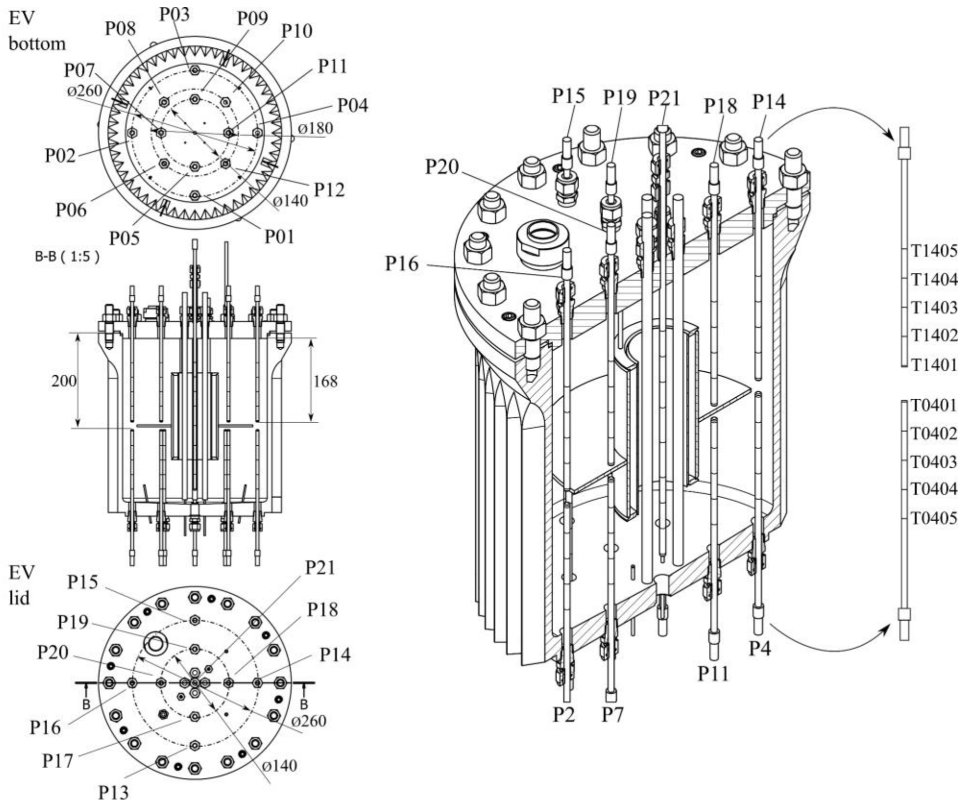

Figure 1: Layout of the temperature sensors in the experimental vessel

Other sensors required for setting and benchmarking of the numerical models are related to the air channel. The inlet air temperature (which corresponds to the ambient temperature) is measured using a platinum resistance thermometer (Pt100) at discharge of the radial fan. The same sensor is placed above the EV for measurement of the outlet air temperature. This sensor is not needed for setting of numerical models but is desirable for control of energy balance. However, it was observed that temperature field on a cross-section of the outlet duct is non-uniform and evaluation of the outlet air temperature might not be precise enough. The air mass-flow rate in the AC is measured using a Wilson flow grid with a PTSXR pressure transducer located over the EV and delivered by the AIRFLOW Company. Based on fundamental principles, it provides a simple, reliable and continuous output. The grid consists of a set of tubes connected by manifolds. The tubes are perforated in such a way as to provide a single differential pressure signal, which is proportional to the square of the mean velocity in the airway. The flow meter therefore considers non-uniform flow distribution in the outlet duct. Moreover, a flow straightener is located in front of the flow meter. The main heaters are connected to DC power supply. Operational range is 0-250 V with max. 40 A current. Used power meter provides accurate determination of the energy flow to the heaters and has own voltage and current measurement. Measured thermal power is the output for setting of a heating boundary condition in a numerical model. Accuracy of instrumentation needed for validation of the numerical models is summarized in Table 1.

Several additional sensors, which are not directly connected with generation of the experimental data were implemented. These sensors are intended to support filling and draining of the EV or for safety reasons. A number of TCs are connected to the auxiliary heaters to control temperature in the SV and the filling pipeline. For the same reasons, TCs were placed directly in the bottom of the EV. Two pressure sensors are used for pressure control in the EV and SV respectively. A contact level sensor connected to the EV lid indicates height of the lead level in the EV. Additional TCs were attached on the external surface of the air channel to provide relevant data for setting of radiation boundary condition in the numerical models.

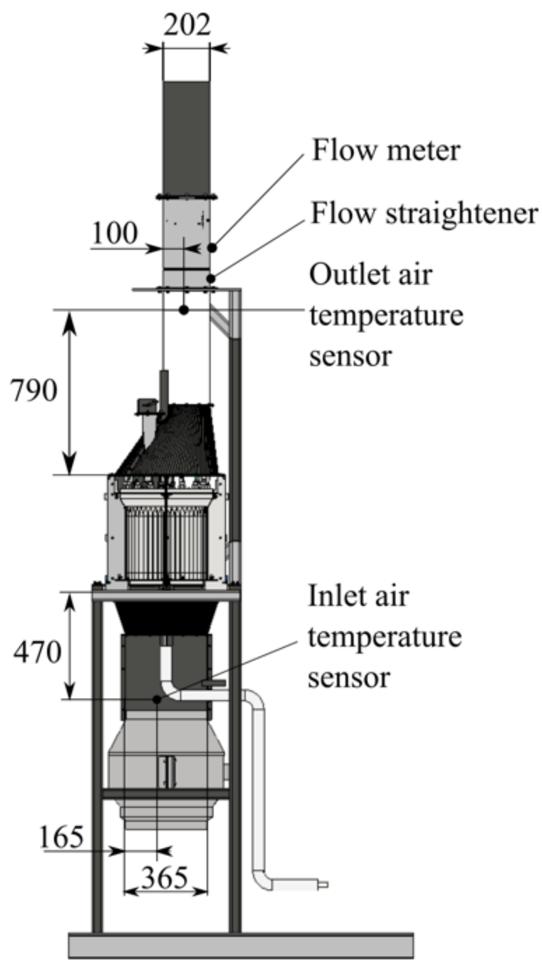

Figure 2: Layout of the air channel

Table 1: Accuracy of sensors needed for models validation

Sensor	Position	Inaccuracy
K-type Class 1 thermocouple	TC probes in the EV	1.5 K below 375°C, 0.4% of the reading above 375°C
Pt100 Class A thermometer	Inlet air temperature	0.15 + 0.002% of the reading
Measuring card	Has to be add to all temperature measurements. Includes aggregate inaccuracy of the trace between the sensor and the display.	0.25 K below 100°C, 0.38 K below 500°C
Power meter	Measurement of the MH power	0.5% of the reading
Wilson grid with PTSXR pressure transducer	Measurement of the air flow rate	5% of the reading

2.4 Experimental campaigns

Each experimental campaign starts from a cold state when all the lead is solid in the SV and both vessel are closed. Both vessels are evacuated and filled with argon several times to remove air. Tightness test is then performed by pressurizing the vessels. After this test, the lead in the SV is heated up. When all the lead is in the liquid state, mixture of argon and hydrogen is pushed in the EV. The mixture reducing amount of oxides passes through the liquid metal and leaves the system through a pipe placed above the lead level in the SV. In parallel with this process, the filling pipeline and the EV bottom might be heated up. When temperature of the system is above the lead melting point, the gas supply is reconnected to pure argon and filling of the EV may start. The filling is provided by increasing the pressure in the SV which pushes the lead up to the EV. When the lead level reaches the contact level sensor, pressure in the FV is hold constant and the draining pipeline is plugged by its sharp cooling. The main heaters are turned on as well as the fan. The experimental campaign then continues by collecting required data according to a predefined experimental matrix. The experiments are typically focused on acquisition of temperature field under steady-state cooling and heating conditions or on transients caused by changes of the operational parameters. Shape of a solid structure can be also obtained by 3D scanning. At the end of the campaign, the EV can be quickly drained leaving the frozen fraction at the bottom of the EV. After that, the experimental setup is cooled down to the ambient temperature, the EV can be opened and the solidified structure is scanned which is the last step of the campaign.

The first experimental campaign that took place in 2018 was focused on measuring of several steady-states represented by various setting of heating power and air mass-flow rate. These data were used for the first assessments of the numerical models (see section 4). The second campaign dealt with fast transients defined by sharp changes of the operational parameters. The third will be focused on both steady-state and transient experiments with modified geometry of the experimental setup. Two modified types of the inner obstacle (an adiabatic obstacle and a steel obstacle with additional ring on the external surface) will be studied. Moreover, experiments with partial blockage of the air channel will be also carried out.

3 COMPUTATIONAL MODELS

Two CFD models were developed during this activity by CVR (using code ANSYS Fluent) and by CRS4 (STAR-CCM+). The models have been gradually modified and improved based on their testing and on findings obtained during the experimental stand operation. The following properties and are demanded of the models:

- Heat transfer and natural convection in liquid lead
- Solidification of lead
- Heat conduction in the solid bodies (the EV, MH, obstacle)
- Accurate modelling of convective heat transfer to the cooling air
- Radiation heat transfer between the external surface of the EV and the external wall of the air channel was found significant and has to be considered as well
- Ability to solve both steady-state and transient cases
- Sufficient accuracy and reasonable computational time.

3.1 Model of CVR

As presence of two fluids with very different properties (air and lead) has been found as a relatively challenging issue, two approaches of the experimental setup modelling were assumed and tested (Melichar 2017). The first approach is based on modelling of the whole experimental section including both lead and the air cooling. This option promises improved accuracy but, at the same time, increased computational time. The second option assumes simplified modelling of the air cooling using a convective boundary condition.

The computational geometry includes two fluid domains (the lead and the air) and several solid bodies. These are the steel EV, steel cylindrical inner obstacle, main heaters and the layer of argon cover gas (modelled as a solid body). The heaters are modelled in details including the steel cover, ceramic insulation and the Kanthal® wire. Several geometric simplifications were adopted. The TC probes are not assumed in the model as well as the chamfer at the bottom. Moreover, the air channel is modelled without the reduction caused by the flange of the EV. One quarter symmetry was assumed.

A hexahedral computational grid was created in the whole computational model. A high quality structured mesh was needed in the lead domain to avoid divergence, unstructured mesh was used for the EV and the air channel. Conformal interfaces between the domains were found necessary due to disturbances occurred during transient simulations. Grid independent results obtained from a mesh sensitivity study were obtained for approx. 4.8 million cells (1.6 M in the lead domain, 2.2 M in the air channel and 1 M in the solid bodies (Iannone 2017). Unstructured tetrahedral or polyhedral grids were found incompatible with used solver settings. Computational geometry can be seen in Figure 3 left, the final mesh with a detail view in the heater is then in Figure 3 right.

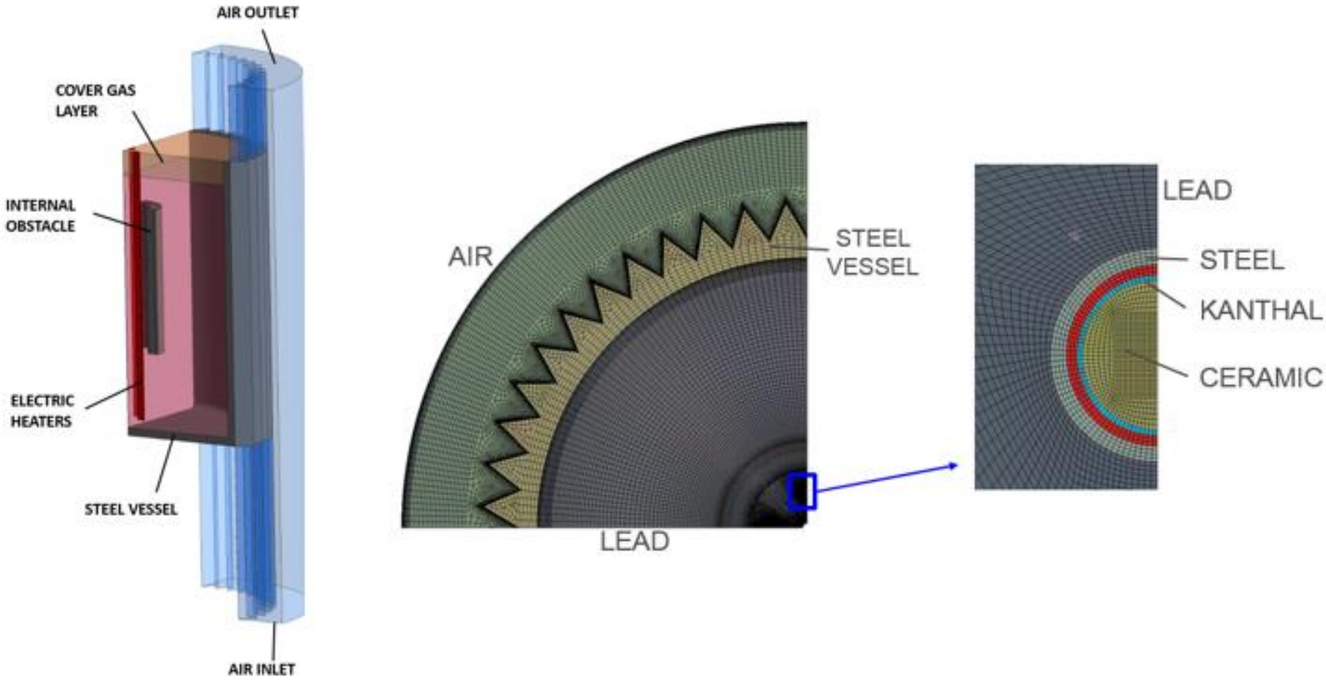

Figure 3: Computational geometry (left), computational mesh (right) of the CVR model

The main features of the numerical model including boundary conditions and the solver settings follows in Table 2:

Table 2: Numerical setting of the CVR model

Boundary conditions (BC)	Heating condition is set as a volumetric heat source in the heating element. “Velocity inlet” characterized by the air inlet velocity and temperature and “Pressure outlet” BC were set in the air channel.
Material properties	Material properties were set as piece-wise linear functions in terms of temperature.
Model of turbulence	Two equations k-Epsilon RNG model, which resolves turbulent Pr number numerically unlike other RANS models was selected. This option allows to handle differences in convective heat transfer in the two fluids. “Enhanced wall treatment” option was used to avoid wall functions in the boundary layer (ANSYS 2013).
Discretization schemes	Second order upwind discretization schemes were used. PRESTO! Scheme was used for pressure discretization as it is recommended for natural convection (ANSYS 2013).
Solidification model	An existing ANSYS Fluent module “Solidification” was employed. The module is based on an enthalpy-porosity technique. A quantity called the liquid fraction indicates fraction of the cell volume that is in liquid form based on the enthalpy balance. Setting of zero porosity to the cells where the material is fully solidified is being applied. The same liquidus and solidus temperature of 600.6 K was set in the lead properties. The latent heat of melting of 23070 J/kg was applied.
Transient setting	An unsteady calculation can be run with relatively high time step (1 s). However, the time step has to be reduced at some phases of the simulation (especially when the frozen front is reaching the obstacle).
Air cooling simplification	As mentioned previously, the option assuming simplified modelling of the air cooling through a convective BC defined by bulk temperature of the fluid and heat transfer coefficient (HTC) is considered. The convection BC is applied on the external surface of the EV while the air channel domain is supressed. The bulk temperature was set as a liner function based on the energy balance. The HTC was estimate using the Dittus-Boelter correlation.

Radiation	The radiation BC implies simplified radiative heat transfer modelling from the external surface of the EV to the air channel cover. This possibility is applicable only if the air channel is suppressed. Emissivity of the material was set to 0.65. The external radiation temperature of 70°C (corresponding to the external wall of the air channel) was considered based on the experimental measurements.

3.3 Model of CRS4

The numerical model prepared using code STAR-CCM+ assumes the geometric features which were neglected in the CVR model such as the TC probes, chamfer at the bottom and the air channel geometry. The substantive difference lies in the meshing approach as the STAR-CCM+ model works with unstructured polyhedral mesh in the whole domain. Number of the mesh elements is 2.8 M. K-Epsilon Realizable model of turbulence with wall treatment was utilized. Unlike the CVR model, the cover gas domain was resolved using "Volume of Fluid" model (VOF). The "Melting-Solidification" model is also accessible in the code through the VOF module (Melichar 2018).

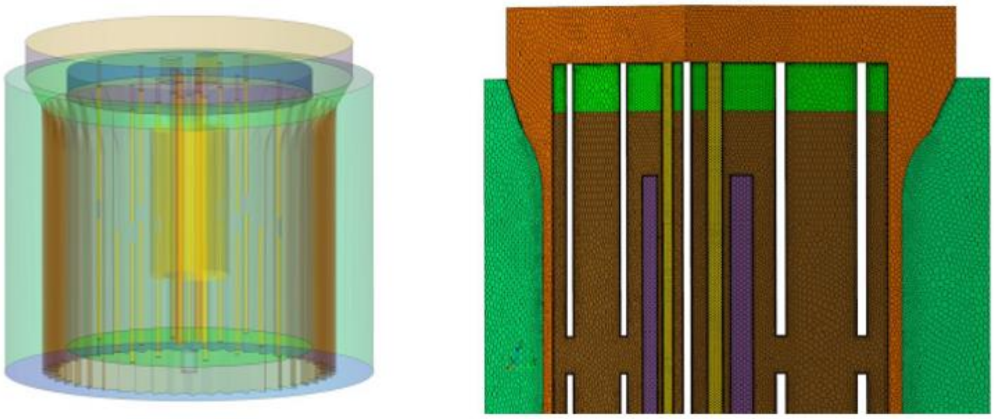

Figure 4: Computational geometry (left) and computational mesh (right) of the CRS4 model (Melichar 2018)

A code-to-code comparison has been already done for pre-test simulations. However, radiative heat transfer has not been included at this phase and therefore results of these pre-test simulations were far away from the experiments (Iannone 2017). In the following section, the steady-state experimental data will be compared with the models supplemented by the radiative heat transfer modelling. The option with supressed air channel in the CVR model was considered for these simulation since the radiation has not been implemented in the full model at this stage.

4 STEADY-STATE EXPERIMENTAL DATA

Four of the steady-state temperature fields measured during the first experimental campaign by nine TC probes in a cross-section of the EV will be presented in this section. Positions of the measuring points correspond to Figure 1. The steady-states were reached by setting of given MH power and the air mass-flow rate. The parameters were then kept constant until the measured temperatures were stabilized.

The experimental results with appropriate operational parameters are shown in Figure 5 to Figure 8 (middle). Moreover, results of the computational models for the same operational conditions were also depicted and can be seen in these picture. Shape of the frozen lead in the experiment might be partially deducted from the TCs with temperature below the melting point (327°C). Percentage of solidified fraction for both models is summarized in Table 3. It is obvious that both models can deal with the natural convection and freezing of the lead in the SESAME-stand facility. It is remarkable to see that the cases depicted in Figure 5 and Figure 8 were better captured with the CVR model while the case in Figure 6 and Figure 7 were better predicted by the CRS4 model. This may mean that the simplified simulation of the air channel as well as the geometric simplification adopted by the CVR model might be acceptable.

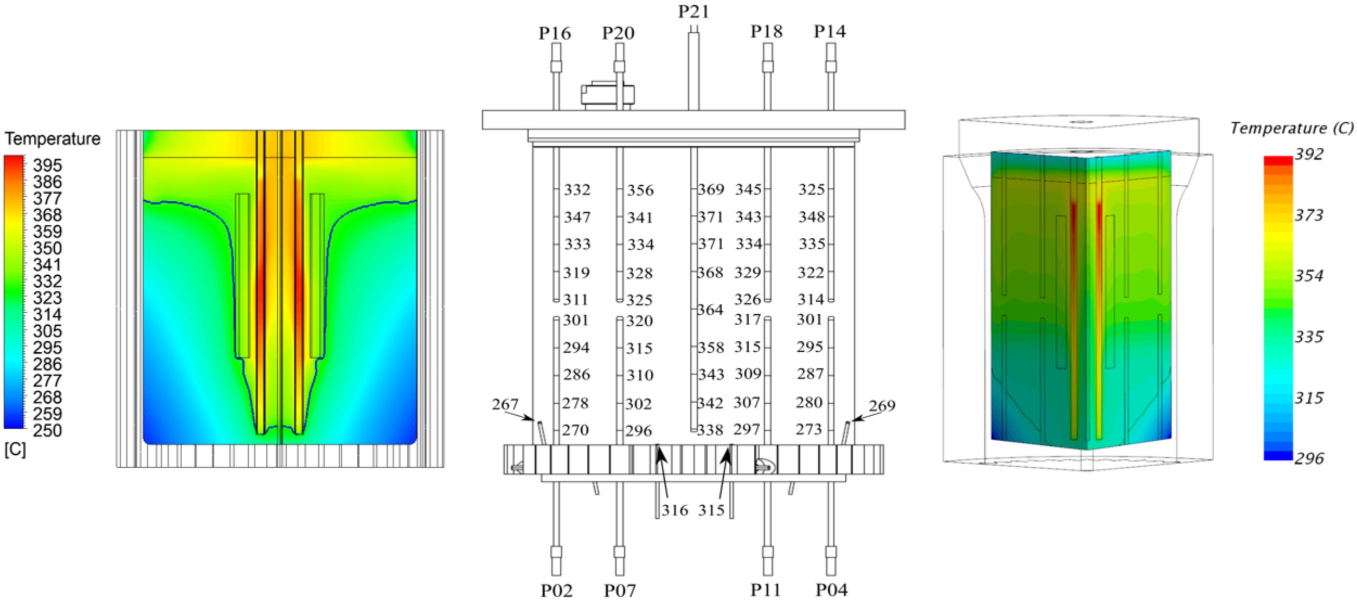

Figure 5: Results of CVR model (left), experimental data (middle) and CRS4 model for 4 kW heaters power and 0.101 kg/s air flow rate

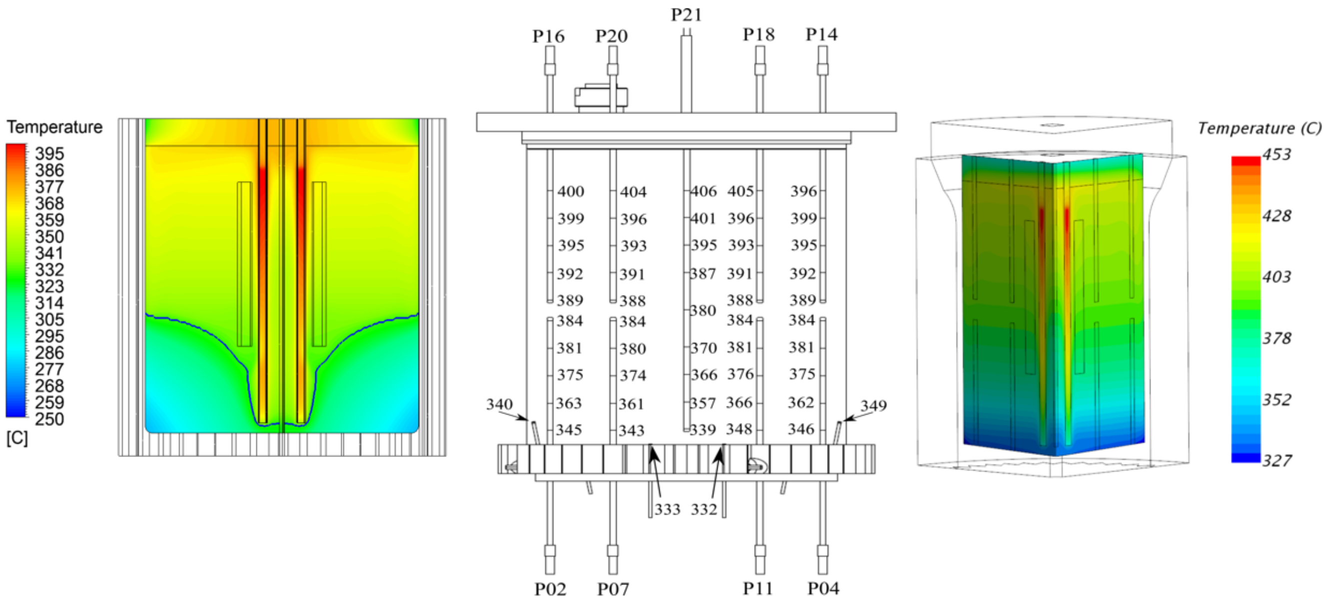

Figure 6: Results of CVR model (left), experimental data (middle) and CRS4 model for 4.96 kW heaters power and 0.101 kg/s air flow rate

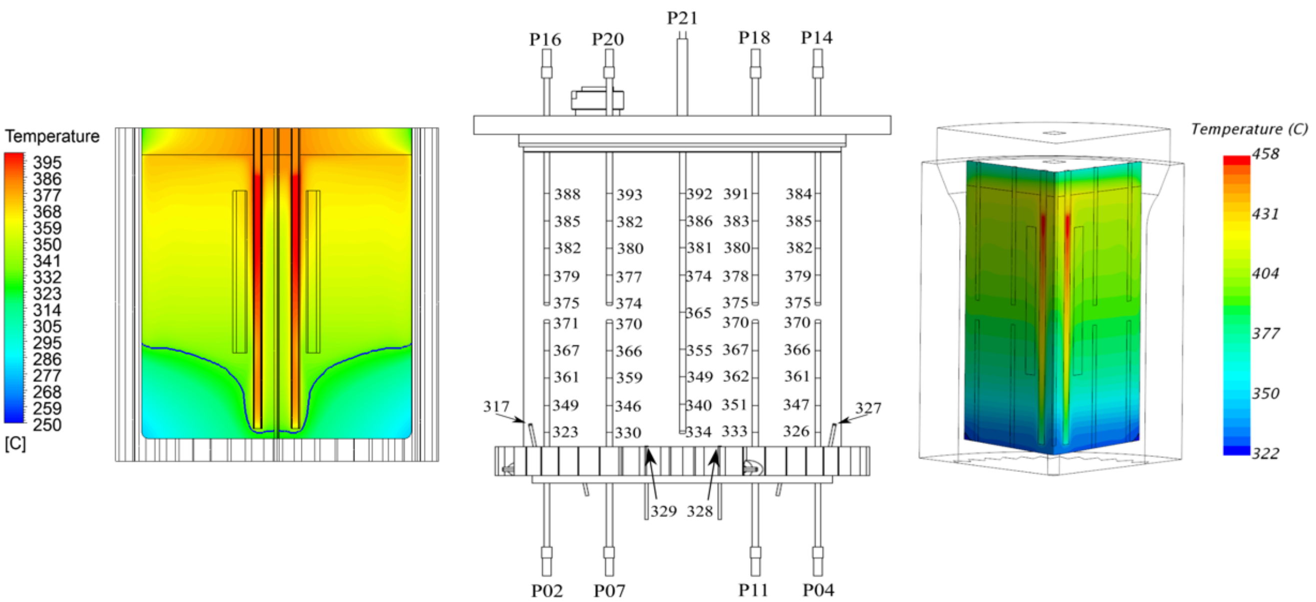

Figure 7: Results of CVR model (left), experimental data (middle) and CRS4 model for 5.6 kW heaters power and 0.145 kg/s air flow rate

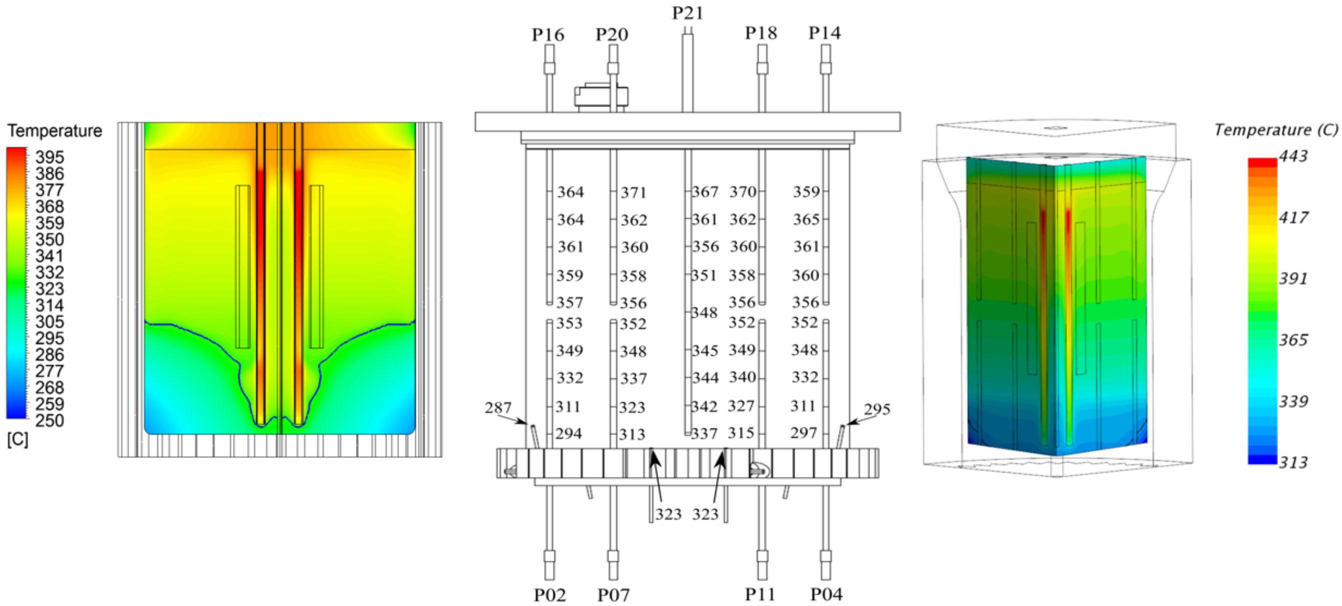

Figure 8: Results of CVR model (left), experimental data (middle) and CRS4 model for 5.6 kW heaters power and 0.167 kg/s air flow rate

Table 3: Comparison of solidified fraction obtained from the CFD models

Case	Solidified fraction (CVR model)	Solidified fraction (CRS4 model)
4 kW, 0.101 kg/s	77,5%	12,5%
4.96 kW, 0.101 kg/s	0%	35,5%
5.6 kW, 0.145 kg/s	0.5%	28,5%
5.6 kW, 0.167 kg/s	1,5%	34%

5 CONCLUSION

The SESAME-stand experimental facility intended for generation of thermal-hydraulic data on lead natural convection and freezing in a pool-type geometry has been designed and operated at CVR. The experimental data might be utilized for validation of numerical models. In this paper, the facility is described in order to provide all parameters required for preparation of a CFD model including geometry, operational parameters and instrumentation. In parallel, two CFD models representing the SESAME-stand were developed using codes ANSYS Fluent and STAR-CCM+ respectively. The models are briefly described in this paper. Selected steady-state temperature fields obtained from the experimental campaign are also presented and the data are compared with the numerical models.

It was found that both models are capable to deal with phenomena related to the SESAME-stand and results of both models are comparable even though the CVR model assumed several simplifications such as modelling of the air cooling through the convection BC or neglecting of some geometric features. However, there is still room for improvement of the models as the differences in the lead temperatures and the solidified fraction are still perceptible. For example, heat loss from the EV caused by radiation was found significant and modelling of this phenomenon should be refined. The lead temperatures are also very sensitive on the air cooling parameters and increased attention needs to be paid to this issue. Apart from the models modifications, experimental methods might be also improved. Additional heat losses through the bottom of the EV, which has not been considered, will be estimated. Moreover, effect of changing of the ambient temperature in the experimental hall were observed which makes achievement of steady-state conditions more difficult.

In the following stage, experimental data focused on transient regimes and modified geometry of the experimental section will be processed. These data as well as shapes of frozen structure obtained using 3D scanning will be available for validation of the numerical models. The computational models mentioned in this paper will be further improved and will be also validated on the experimental data.

ACKNOWLEDGEMENT

The authors would like to thank to the European Commission for its financial support within the HORIZON2020 SESAME program (n^0 654935).

The presented work was financially supported by the Ministry of Education, Youth and Sport Czech Republic - project LQ1603 Research for SUSEN. This work has been realized within the SUSEN Project (established in the framework of the European Regional Development Fund (ERDF) in project CZ.1.05/2.1.00/03.0108 and of the European Structural and Investment Funds (ESIF) in the project CZ.02.1.01/0.0/0.0/15_008/0000293).

NOMENCLATURE

AC	Air Channel
BC	Boundary Condition
CFD	Computational Fluid Dynamics
CVR	Research Centre Rez
CRS4	Center for Advanced Studies, Research and Development in Sardinia
EV	Experimental Vessel
MH	Main Heaters
SESAME	Thermal hydraulics Simulations and Experiments for the Safety Assessment of Metal cooled reactors
SV	Storage Vessel
TC	Thermocouples
VoF	Volume of Fluid
WP	Work Package

REFERENCES

ANSYS 2013 ANSYS Fluent Theory Guide Release 15.0, ANSYS, 2013
Iannone M. et al. 2017 Development of CFD models and pre-test-calculations for thermal-hydraulics and freezing experiments on Lead coolant, NENE2017, conference paper.
Melichar T. et al. 2017 CFD Modeling of Natural Convection and Freezing Phenomena for Heavy Liquid Metal Coolants, In Nuclear Technologies for the 21st Century; Sep 13, 2017; ISBN: 978-80-270-3233-4.
Melichar T. et al. 2018 Pre-test Simulations and Measurements from the Meliloo Test Facility, deliverable of the H2020 SESAME project (internal report)
Moreau V. et al. 2019 Pool CFD Modelling: Lessons from the SESAME Project, SESAME International Workshop contribution, this conference.