FLOW BLOCKAGE THERMAL-HYDRAULIC ASSESSMENT IN ALFRED FUEL ASSEMBLY

R. MARINARI[1], I. DI PIAZZA[1], M. TARANTINO[1], D. MARTELLI[1]

[1] ENEA, Department of Fusion and Nuclear Safety Technology, 40032, Camugnano (BO), Italy.

Email contact of corresponding author: ranieri.marinari@enea.it

EXTENDED ABSTRACT

The Advanced Lead-cooled Fast Reactor European Demonstrator (ALFRED) is a 300 MWth pool-type reactor aimed at demonstrating the safe and economic competitiveness of the Generation IV LFR technology. The ALFRED design, currently being developed by ANSALDO NUCLEARE and ENEA in the frame of the FALCON Consortium, is based on prototypical solutions intended to be used to boost the DEMO-LFR development.

In the frame of the research activities devoted to ALFRED development, the flow blockage in a fuel sub-assembly is considered one of the main issues to be addressed. The flow blockage accident consists of a partial or total occlusion of the flow passage area. This can lead to a degradation of the heat transfer and mass flow rate in a fuel assemby, potentially causing a temperature peak in the clad which can lead to the degradation of the clad barrier. The blockage phenomenon in lead cooled bundles was preliminarly investigated in the past by CFD numerical analysis documented in [1].

This work reports the experimental results and post-test analysis carried out in the prototypical test section of the lead-bismuth eutectic (LBE) -operated NACIE-UP facility. NACIE-UP is a rectangular loop, where the two vertical pipes, working as riser and downcomer, are 8 m long and the two horizontal pipes are 2.4 m long; for a more detailed description of the facility and for typical experiments please refer to [2].

The heat source called Blockage Fuel Pin Simulator (BFPS) consists of 19 electrically heated pins with an active length of 600 mm and a diameter D= 10 mm. The pitch to diameter ratio is P/D=1.4. The maximum external pin heat flux is ≈0.7 MW/m^2. The pins are placed on a hexagonal layout by a suitable wrapper, while two spacer grids maintain the pin bundle in the correct position. The total power of the pin fuel bundle is 250 kW.

This fuel pin bundle configuration is relevant for the thermal-hydraulic design of the ALFRED core.

The BFPS is installed in the bottom of the riser, whereas a shell and tubes heat exchanger (heat sink) is placed in the upper part of the downcomer. The main components layout allows the facility to work both in forced and natural circulation regimes. Different internal blockages were simulated inside the BFPS test section with 10% to 33% blocked area ratios (FIG. 1-upper), blockage 2 was a blockage around the central pin but it was not performed in the experimental campaign due to its irrelevan flow area obstruction. The degree of blockage can be fixed by moving appropriate rods in the bottom part of the test sections that obstract some of the spacer grid holes with a small shaped plate. A plate obstruction is placed at the bottom of the active region on the first spacer grid, same as the experiment in in [1] and in a preliminary CFD pre-test analysis documented in [3]. According to preliminary results of the numerical pre-test analyses, the thermocouples were mainly positioned in the first 100 mm downstream of the blockageand along the active region.

Experimental data on various degrees of blockage showed a maximum temperature closer to the blockage 30 mm downstream the obstacle. The peak temperature value is around 25-45 °C higher than the inlet LBE temperature (about 200°C) in the different experimental conditions and different locations, see FIG. 1-lower. Although the blockage phenomenology is clear and it is repeated in each

condition, many data were produced to be used for the validation of CFD codes and numerical methods applied to internal blockage in grid-spaced fuel assembly.

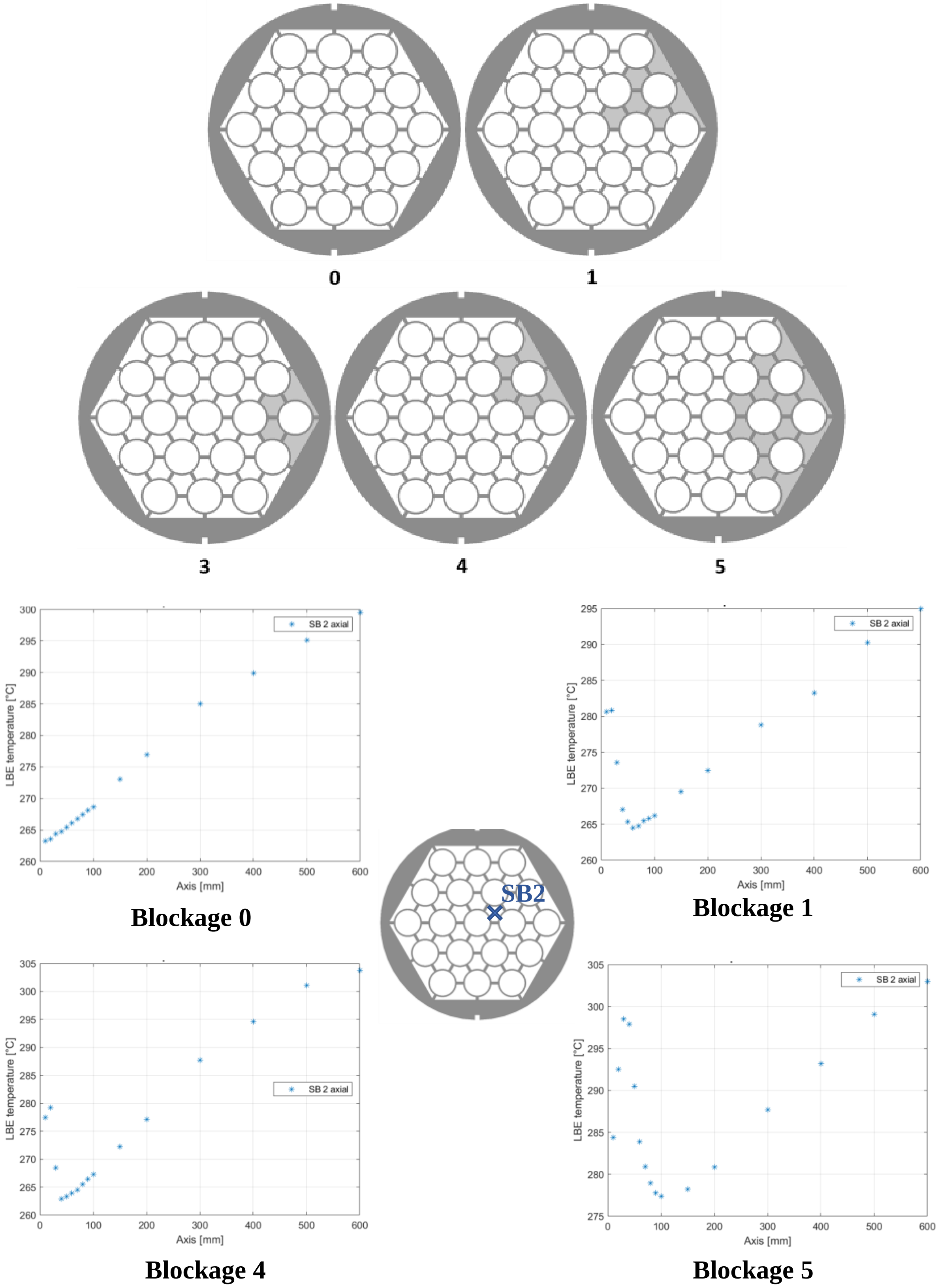

FIG. 1. Simulated blockages (upper) and experimental axial temperature profiles along a sub-channel (lower).

A CFD numerical post-test validation activity was carried out on a limited number of unblocked and blocked cases. The CFD numerical model developed reproduces the geometry of the test section in a detailed way. An appropriate mesh sensitivity study was performed looking for most accurate and less computationally expensive model (FIG. 2). Different numerical models were tested with RANS and URANS simulations. For the single sector blockage numerical and experimental results are compared in detail; for a more extended description see [4].

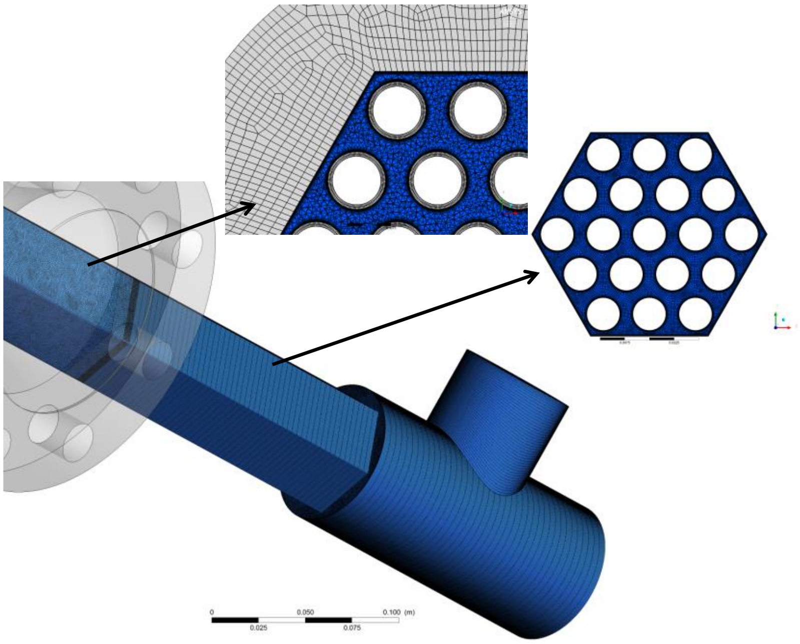

FIG. 2. General sketch of the computational domain developed. Structured mesh under the blockage and unstructured mesh over it are pointed out. Pins and pipe mesh are shown together with the fluid one.

Results show that the CFD code adopted (ANSYS CFX) is capable to predict the location of the maximum peak temperature and this feature is also evidenced by experimental datas, although the quantitative comparison is not always satisfactory, see for example FIG. 3.

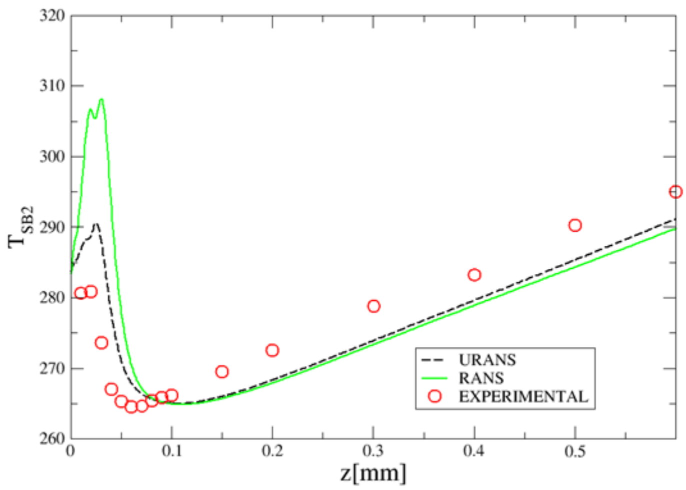

FIG. 3. Experimental vs. numerical temperature distribution on the central subchannel for sector blockage case (1/6 of the flow area).

Both numerical and experimental results show two separated main effects of the blockage: a local effect with a maximum in temperature field behind the blockage (first peak due to the vortex recirculation downstream the blockage) and an overall effect with a local maximum at the end of the active region in the blocked subchannels (due to the lower mass flow rate in the bundle). These two effects were originally found in [1] and have been confirmed by experimental data and further numerical simulations. The comparison of experimental and numerical data shows that an unsteady RANS simulation provides a lower temperature peak in the recirculation region downstream of the blockagein better agreement with the experimental data; anyway, the width of the temperature peak is similar to the steady state RANS. The better agreement of the unsteady RANS simulation indicates that a more accurate approach like a LES simulation could reduce the difference with the experimental results.

ACKNOWLEDGEMENTS

This work was performed in the framework of H2020 SESAME project. This project has funded by the European Commission under grant agreement No 654935.

REFERENCES

[1] DI PIAZZA, I., MAGUGLIANI, F., TARANTINO, M., ALEMBERTI, A.," A CFD analysis of flow blockage phenomena in ALFRED LFR demo fuel assembly" Nuclear Engineering and Design **276** (2014) 202-215.

[2] DI PIAZZA, I., ANGELUCCI, M., MARINARI, R., TARANTINO, M., MARTELLI, D., "Thermo-fluid dynamic transients in the NACIE-UP facility" Nuclear Engineering and Design **352** 110182 (2019).

[3] MARINARI, R., DI PIAZZA, I., FORGIONE, N., MAGUGLIANI, F., "Pre-test CFD simulations of the NACIE-UP BFPS test section" Annals of. Nuclear Energy **110** (2017) 1060-1072.

[4] MARINARI, R., DI PIAZZA, I., TARANTINO, M., FORGIONE, N., "Blockage fuel pin simulator experiments and simulation" Nuclear Engineering and Design **353**, 110215 (2019).