Application of CFD and System Codes to Simulation of HERO-2 Experiment

Adam Kecek
UJV Rez, a. s.
Hlavni 130
250 68, Husinec-Rez, The Czech Republic
adam.kecek@ujv.cz

Ladislav Vyskocil
UJV Rez, a. s.
Hlavni 130
250 68, Husinec-Rez, The Czech Republic
Ladislav.vyskocil@ujv.cz

ABSTRACT

The EU PASTELS project aims at demonstration of how innovative passive safety systems can support modernisation and optimisation of the European nuclear industry. Within the project, several passive heat removal experiments are studied both with system and CFD codes. One of them is the HERO-2 experiment, which simulates heat removal from bayonet tubes through piping and condenser submerged in water pool. In this paper, both open extended test and several closed loop tests are used for evaluation of the code capabilities. The CFD simulations are conducted with Neptune_CFD and Ansys CFX codes, where different setup and modelling approaches will be discussed. System code simulations are done with ATHLET 3.3, which is a part of the AC2 code package. The summary discusses existing challenges and drawbacks of the applied approaches and suggests areas of future code development.

1 INTRODUCTION

Safety of nuclear reactors during variety of accidents relies on heat removal systems. Previously, these systems were typically active. The modern Gen III+ reactors and numerous Gen IV and SMR concepts incorporate passive heat removal systems, where the driving force is purely natural. The European research project PASTELS (PAssive Systems: Simulating the Thermal-hydraulics with Experimental Studies) focuses on demonstration of benefits of the passive heat removal systems in nuclear industry. Within this project, several experimental programmes and numerical activities are conducted investigating the capabilities of both CFD and system codes in different situations. One of these experiments is the HERO-2 facility.

The numerical activities in this paper aim at application of ATHLET, Neptune_cfd and Ansys CFX on simulation of the open extended test with forced flow and on some tests in closed loop configuration. The scope of these activities is to test the applicability of different computational approaches in numerical simulation of passive safety systems.

2 FACILITY DESCRIPTION

The HERO-2 experimental facility [1], [2] is a part of the studies conducted for the design of steam generator systems for SMR light water reactor applications, to remove the decay heat generated by an emergency shutdown through natural circulation. Thanks to the modifications made to the system in previous experimental campaigns, the test section can be tested both in open circuit forced circulation, fed with subcooled or saturated water, or in closed circuit natural circulation.

The 20 m high facility consist of following main parts. Two bayonet tube steam generators, a hot leg connecting the test section outlet to the pool condenser inlet, a slightly inclined pool condenser pipe submerged in a condensation pool for heat removal, a cold leg connecting the pool condenser outlet to the test section inlet.

Each of the two bayonet tubes is made of three concentric tubes. The central tube provides inlet of the coolant. At the inlet of each tube, an orifice is placed for water flow stabilisation. The sealed gap between the inner and central tube serves as insulation between the flow channels to reduce the thermal flux to the downward flow. The outer tube is electrically heated on the outer side. Both bayonet tubes end in a steam chamber, which is connected to the hot leg. All piping and tubes are insulated with rockwool. The scheme of the facility is presented in Figure 1.

The facility can be operated in several configurations. The closed loop configuration is used for tests with natural circulation. In this case valves V18 and V20 are closed, while V19 is open. The open extended configuration is used to determine dispersions in different sections of the circuit, i.e. pressure loss coefficients estimation etc., with forced flow of subcooled water. In this case, whole facility except small section in cold leg providing coolant inlet outlet can be tested.

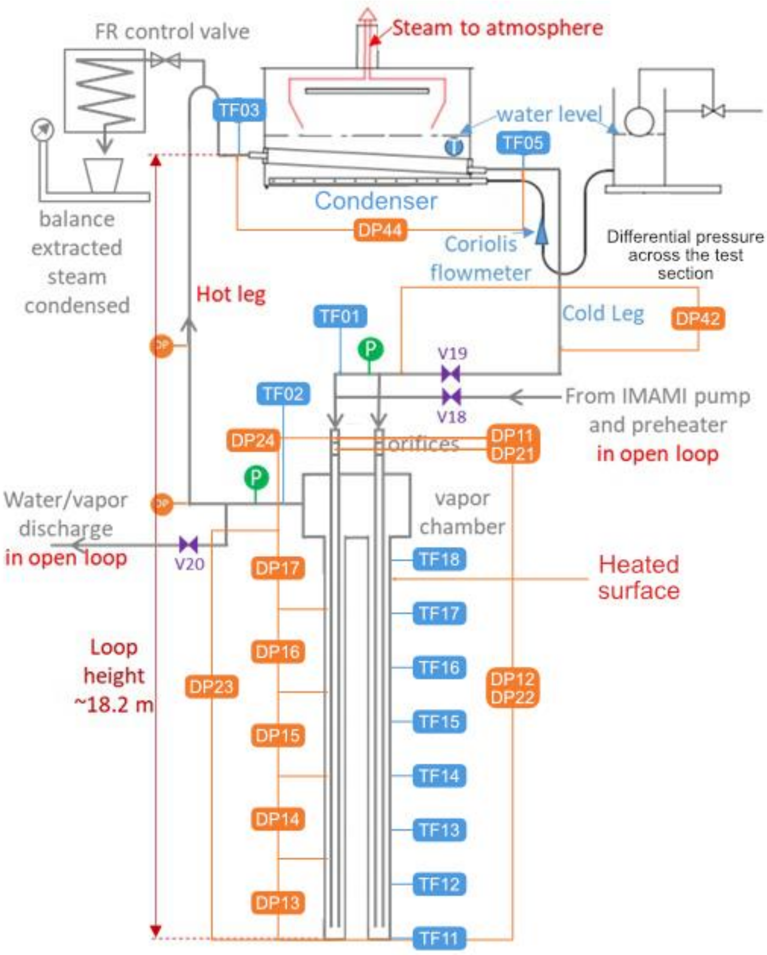

Figure 1: HERO-2 Experimental facility [1]

3 SIMULATION AND RESULTS

3.1 ATHLET simulations

The simulations were conducted with ATHLET 3.2 [3], which is a part of the AC2 2019 code package developed by GRS, gGmbH. To simulate the experiment, both closed loop and open extended loop were modelled.

Both open extended and closed loop nodalisations were developed. Vast of the volumes are defined as pipe thermo-fluid objects (TFOs). A branch object is used to model the vapour chamber collecting the evaporated coolant from the bayonet tubes. For the open extended test, time dependent volume and FILL object were added to simulate the inlet and outlet of the experimental facility. Furthermore, the model includes heat structures, which are composed of steel and rockwool, to simulate the piping and insulation respectively. GCSM control signals were added as well, to simulate boundary conditions.

Open extended test simulation in ATHLET

The simulation of open extended test aimed at proper setup of the initial and boundary conditions, i.e. pressure and heat losses. The heat losses of the insulated piping were modelled by constant heat transfer coefficient (HTC). Similar approach was adopted to simulate the heat transfer on the outer side of the condenser pipe in pool. The heat transfer in the gap between the bayonet tubes was simulated by prescribed heat transfer coefficient as well to incorporate heat transfer due to convection and radiation. The list of the obtained boundary conditions is presented in Table 1. Regarding the results, good agreement was obtained both for temperatures and pressures. Significant difference obtained for thermocouple TF15 may be neglected due to malfunction of this device. The comparison of calculated results and experimental values is presented in Figure 2.

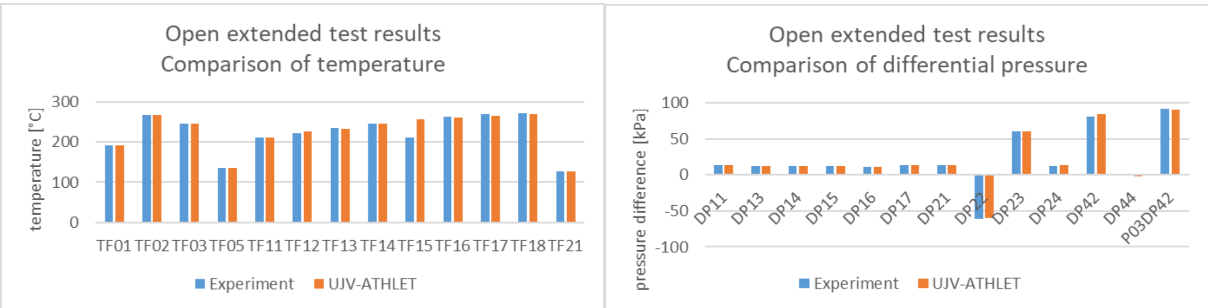

Figure 2: Comparison of experimental data and ATHLET calculation results of the Open extended test Left: Temperature along bayonet tube. Right: Differential pressure in the loop.

Closed test simulations in ATHLET

Closed tests No. 1 - 3, 7 and 8 were simulated with ATHLET. Simulation of the Test 1 with initial and boundary conditions obtained from simulations of the open extended test revealed significant discrepancies compared to the experimental data. To achieve a better agreement with the experimental data, the boundary conditions were modified accordingly. The idea was to fit the system pressure, temperature, and filling ratio. After tuning the boundary conditions, the results were in better agreement. The modified boundary conditions are presented in Table 1. The temperatures in the bayonet tube exhibited similar trend and values, except TF15, which was broken. Differential pressures measured alongside the loop exhibit similar trend with larger discrepancies. The results are presented in Figure 3. The calculated mass flow was overestimated by more than 21 %.

Table 1: List of boundary conditions used for open extended test and test 1

Boundary condition	**Open extended**	**Closed test 1**
Heat power	87.7 % Nominal power	73 % Nominal power
Cold leg HTC	6.1 $W.m^{-2}.K^{-1}$	12.1. $W.m^{-2}.K^{-1}$
Hot leg HTC	7.7 $W.m^{-2}.K^{-1}$	7.214 $W.m^{-2}.K^{-1}$
Condenser HTC	2 600 $W.m^{-2}.K^{-1}$	1 700 $W.m^{-2}.K^{-1}$
Gap HTC	77 $W.m^{-2}.K^{-1}$	118 $W.m^{-2}.K^{-1}$

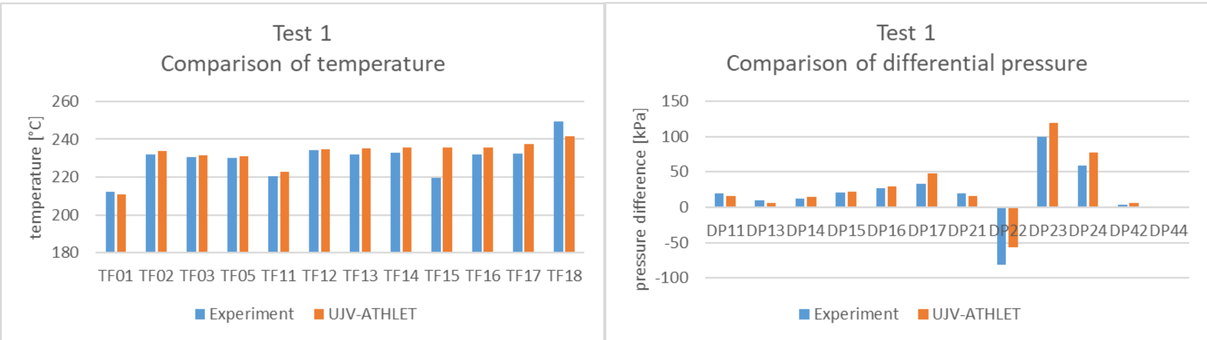

Figure 3: Comparison of experimental data and ATHLET calculation of the Closed Test 1 Left: Temperature along bayonet tube. Right: Differential pressure in the loop.

For the remaining tests, the boundary conditions had to be adapted case by case. In general, the heating power of the bayonet tube was between 60 and 90 %. The source of this discrepancy is obvious. The model does not incorporate heat loss on the outer wall of the bayonet tube. This yields to the need of reducing the power of the heater to incorporate this heat loss. Significant changes were done for the cold leg HTC, which ranged from 7 in Test 7 to 18 in Test 8. The source of this problem is unknown. Remaining boundary conditions were similar to those used in the closed Test 1 simulation.

Regarding the results of other tests, good agreement was observed for temperatures, where the differences reached unit of percent typically. Different situation was achieved for the differential pressures. The trends were in good agreement generally, but the difference between experimental and calculated values were larger but did not exceed the order of magnitude.

A separate problem, which is tied with pressure losses is the mass flow through the circuit. In all cases, the mass flow was overestimated significantly. The difference ranges from 21 to 60 %.

3.2 Neptune_cfd simulations

Simulations performed with Neptune_cfd 6.2 code are presented in this chapter. Neptune_cfd [4] is the multiphase-flows solver build upon the Code_Saturne [5] framework. It is based on a multi-fluid Eulerian approach. It is developed within the framework of the Neptune project, a joint R&D initiative for the development of the next-generation simulation tools for nuclear thermal-hydraulics involving companies CEA, EDF, FRAMATOME, and IRSN.

Computational domain for the Neptune_cfd code covers whole loop: 2 bayonet tubes, vapor chamber, hot leg, primary side of condenser and cold leg. For the simulation of the open extended test, a small part of the cold leg is removed from the domain. Solid volumes (heat structures) are not included in the computational domain. Mesh contains 3.5M cells, most of them hexahedral.

Open extended test simulation in Neptune_cfd

The following assumptions were used. Simulation is performed with single-phase incompressible flow. Heat conduction in solid volumes is not modelled. Orifices were replaced with porous zone with momentum loss terms. To achieve the temperature TF02 at the vapor chamber outlet, the heating powers of both tubes are decreased to 88.7% of nominal value. The rest of heating power goes to heat losses. Heat losses from the cold leg wall and chamber wall are modelled with ambient temperature and external heat transfer coefficient. To achieve the temperature TF11 at the bottom of the bayonet tube, wall heat fluxes into inner tubes were adjusted. The same power is removed from the inner wall of the annular channel of the bayonet tube. In both cases, constant wall heat flux along the vertical coordinate is assumed. The temperature TF03 at the condenser inlet is achieved imposing the heat losses in the hot leg wall, modelled with ambient temperature and external heat transfer coefficient. The temperature TF05 at the condenser outlet is achieved imposing the heat removed from the condenser, modelled with outside temperature and external heat transfer coefficient.

Turbulence is modelled with SST k-omega model. Physical properties of liquid water are modelled with the CATHARE tables.

Neptune_cfd 6.2 was able to correctly predict the pressure losses along the loop in the Open extended test with single-phase flow (Figure 4).

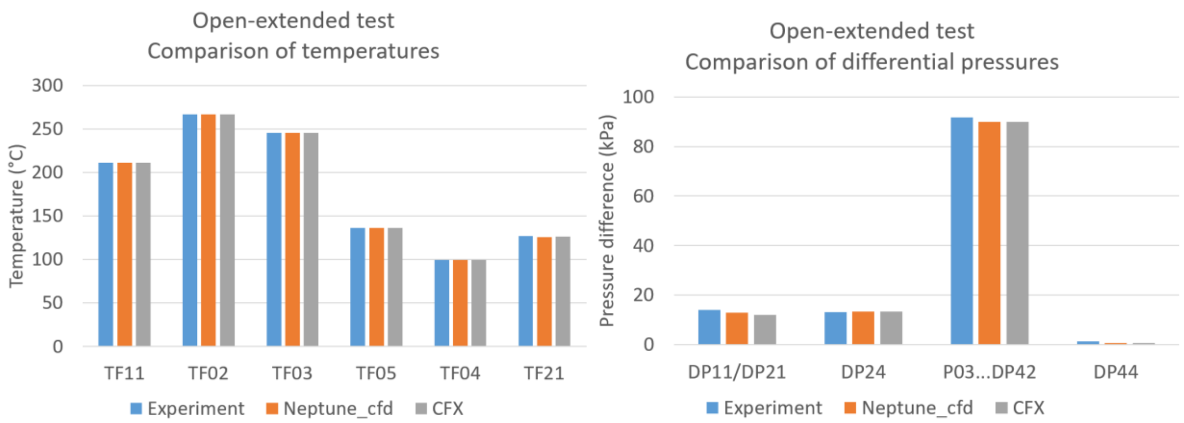

Figure 4: Comparison of experimental data and results of the Open-extended test obtained with CFD codes. Left: Temperatures along the loop. Right: Differential pressures in the loop.

Closed loop tests simulation in Neptune_cfd

Simulation is performed with the Eulerian multiphase model with the Generalized large interface model (dispersed bubbles and large interface). Turbulence is modelled with Rij-epsilon EBRSM model. Both phases, water liquid and water vapour are compressible. Exact value of heat losses from the heater is not known from the experiment. It was assumed that ratio of heat losses to nominal heater power is the same as in the open extended test. Physical properties of liquid and steam water are modelled with the CATHARE tables. The solver settings are based on advice from neptune_cfd code developers from EDF.

At first, individual components have been simulated, bayonet tube and condenser, under the conditions of the closed tests. After that, we simulated one test in the whole closed loop.

Neptune_cfd can simulate boiling in bayonet tube under the conditions of HERO-2 closed loop tests. The results indicate that the modelling approach used in our simulations might have problems with prediction of pressure loss in channel with boiling flow. It is possible that the relative heat losses in closed loop tests are larger than in the Open extended test.

Neptune_cfd can simulate condensation in condenser under the conditions of HERO-2 closed loop tests. Very thin cells (0.1 mm) in the mesh on the condensing wall are needed to reproduce the condensation.

Simulation of the whole closed loop is extremely numerically expensive. After one month computer run time on 400 processors, 31 s of transient time was simulated which was not enough to reach the steady-state conditions in the loop. It is estimated that several hundreds of seconds of transient would be needed to reach steady state. Such a simulation could not be performed in reasonable time at computers available at UJV. Nevertheless, the results obtained might be viewed as a demonstration of the capabilities of the code. Examples of results are shown in Figure 5 and Figure 6.

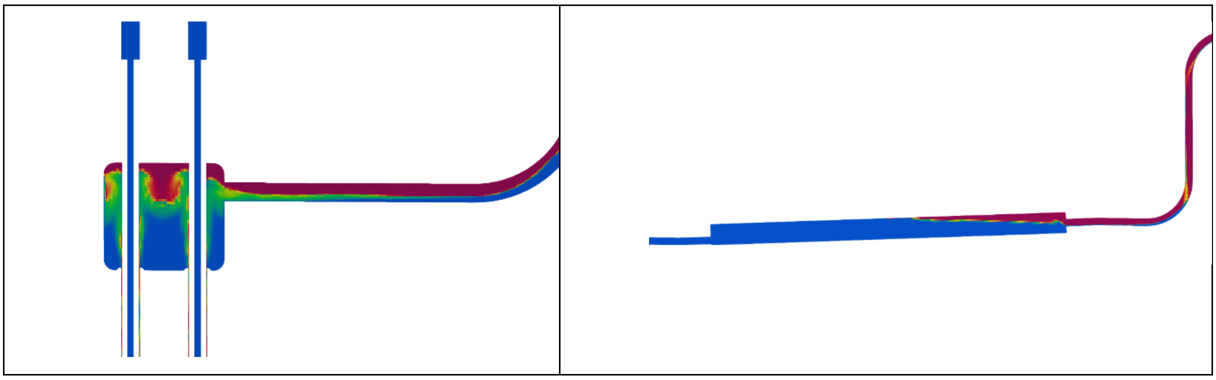

Figure 5: Simulation of closed loop test No. 8 in Neptune_cfd. Left: vapor volume fraction [-] in the vertical cross section through the vapor chamber. Right: vapor volume fraction [-] in the vertical cross section through the condenser

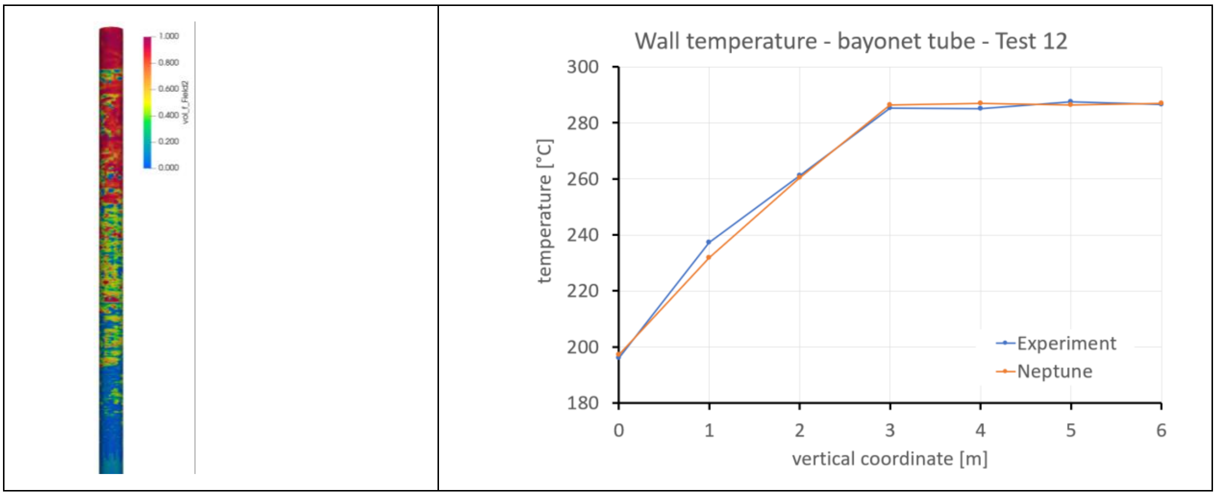

Figure 6: Simulation of individual bayonet tube under conditions in closed loop test No. 12 in Neptune_cfd. Left: vapor volume fraction [-] at the end of the bayonet tube. Right: wall temperature along the height of bayonet tube, simulation and experiment [-]

3.3 Ansys CFX simulations

Here we present simulations with Ansys CFX 18 code. Ansys CFX [6] is a is commercially available general purpose CFD software suite that combines an advanced coupled solver with powerful preprocessing and postprocessing capabilities.

Computational domain covers whole loop: 2 bayonet tubes, vapor chamber, hot leg, primary side of condenser and cold leg. For open extended test, small part of the cold leg is removed from the domain. Inner tube, slave tube in bayonet tubes and air between them are modelled as solid volumes. Orifices are replaced with porous zones. The mesh in fluid domain is nearly the same as the mesh used in Neptune_cfd code simulations in chapter 3.2. Mesh contains 6.1M cells, most of them hexahedral.

Open extended test simulation in Ansys CFX

The following assumptions were used. Simulation is performed with single-phase flow. To achieve the temperature TF02 at the chamber outlet, the heating powers of both tubes are decreased to 88.7% of nominal value. The rest of heating power goes to heat losses. The heat losses from the cold leg wall and chamber wall are modelled with ambient temperature and external heat transfer coefficient. To achieve the temperature T11, the air conductivity was artificially increased to account for radiation heat transfer in air gap. To achieve the temperature TF03 at the condenser inlet, heat losses from the hot leg wall are modelled with ambient temperature and external heat transfer coefficient. To achieve the temperature TF05 at the condenser outlet, heat removed from the condenser is modelled with outside temperature and external heat transfer coefficient.

Turbulence is modelled with SST k-omega model. Physical properties of liquid water are modelled with the IAPWS-IF97 tables.

Ansys CFX 18 was able to correctly predict the pressure losses along the loop in the Open extended test with single-phase flow (Figure 4).

Closed loop tests simulation in Ansys CFX

Many attempts were made to simulate closed loop tests No. 8 and No. 12 in CFX 18 code. Euler-Euler model with generalized wall boiling model and surface tracking model were used in the simulation of two-phase flow. All attempts were unsuccessful. The code always crashed, and it could not provide a converged solution. We also tried to simulate closed tests No. 8 and No. 12 with CFX 18 in open loop configuration. Results with full heating power could not be achieved.

4 SUMMARY

Experimental and numerical activities play essential role in improving the nuclear safety and reliability. The demand on passive safety is growing, so proper understanding the phenomena and appropriate modelling becomes more and more important. The activities summarized in this paper revealed the possibilities of using CFD and system code approach in simulating the HERO-2 facility.

Regarding the system codes, the ATHLET simulation exhibited very good agreement for the open extended configuration with forced flow. In the closed loop tests with natural convection, good agreement was obtained for the temperature profile in the bayonet tube. Significant difference was observed for pressure differences as well as for the mass flow, which was overestimated in all cases.

As regards of the CFD codes, open-extended test with forced flow can be simulated in both Neptune_cfd 6.2 and Ansys CFX 18 codes without problems. Neptune_cfd can be used to simulate individual components of the loop, bayonet tube and condenser, under the conditions of closed-loop tests. Simulation of the whole closed loop with natural circulation is possible in Neptune_cfd but extremely numerically expensive, steady state solution could not be obtained on our computer in time available. Ansys CFX 18 could not be used for the simulation of the closed-loop tests. The code always crashed, and it could not provide a converged solution. The source of this problem is unknown and will be studied in future.

ACKNOWLEDGMENTS

The PASTELS project has received funding from the Euratom research and training programme 2019-2020 under grant agreement No 945275. This text reflects only the author's view and the Commission is not responsible for any use that may be made of the information contained therein.

The PASTELS project idea has also obtained the NUGENIA label on 23/09/2019 (Certificate number: 2019NUG0076) prior to the submission of the project to the European Commission.

REFERENCES

[1] M. Polidori, P. Meloni, C. Lombardo, G. Bandini, M.E. Ricotti, A. Achilli, O. De Pace, D. Balestri, G. Cattadori, “Natural circulation test campaign on HERO-2 bayonet tubes test section”, Proceeding of IAEA Conference Topical Issues in Nuclear Installation Safety, 2017.

[2] M. Polidori et al.: PASTELS Deliverable 2.2: Description of HERO-2 facility and simulations, R1.0. European Commission NFRP-2019-2020-04 Innovation for Generation II and III reactors. 20/01/2021.

[3] Lerchl, G. et al. 2019. ATHLET 3.2 – User’s Manual, distributed with ATHLET, GRS, March 2019.

[4] EDF, NEPTUNE CFD version 6.2. User guide, EDF R&D Fluid Dynamics, Power Generation and Environment Department, Chatou, France, February 1, 2022.

[5] https://www.code-saturne.org/cms/web/NEPTUNECFD [Online].

[6] ANSYS CFX-Solver Theory Guide, ANSYS, Inc. Release 18.0, January 2017, http://www.ansys.com.