NURETH14-492

GENERAL OVERVIEW
OF THE TOPFLOW-PTS EXPERIMENTAL PROGRAM

P. Péturaud[1], U. Hampel[2], A. Barbier[3], J. Dreier[4], F. Dubois[5], E. Hervieu[6], A. Martin[1] and H.-M. Prasser[7]
[1] Electricité de France, France
[2] Helmholtz Zentrum Dresden-Rossendorf, e.V., Germany
[3] AREVA, France
[4] Paul Scherrer Institut, Switzerland
[5] Institut de Radioprotection et de Sûreté Nucléaire, France
[6] Commissariat à l'Energie Atomique, France
[7] Swiss Federal Institute of Technology (ETHZ), Switzerland
pierre.peturaud@edf.fr, u.hampel@fzd.de, anthony.barbier@areva.com, joerg.dreier@psi.ch, franck.dubois@irsn.fr, eric.hervieu@cea.fr, alain-cc.martin@edf.fr, prasser@lke.mavt.ethz.ch

Abstract

During a hypothetical loss-of-coolant accident, cold emergency core cooling water may be injected in a partially uncovered cold leg, and related two-phase CFD simulations are required to demonstrate the reactor pressure vessel integrity further to the resulting pressurized thermal shock. To complement the physical validation of the two-phase CFD codes in this configuration, a dedicated integral-type experimental program – TOPFLOW-PTS – has been setup; this paper (i) supports the validation data needs, (ii) presents the related experimental setup, and (iii) provides the general guidelines of the test matrix definition and test procedure.

1. Introduction

During a hypothetical Loss-Of-Coolant Accident (LOCA) in a Pressurized Water Reactor (PWR), Emergency Core Cooling (ECC) cold water is injected into the cold leg and partially mixes with the hot primary fluid flow down to the downcomer. Such a scenario may result in large Pressurized Thermal Shocks (PTS) on the structural components, first of all the Reactor Pressure Vessel (RPV) wall; and the concern is to reliably calculate the temperature fields in the RPV wall, so that its structural behaviour can be predicted and the associated margins with respect to its loss of integrity assessed. It is an actual today issue in view of the plant aging and life time extension; to meet it, the use of sophisticated codes based on Computational Fluid Dynamics (CFD) is required for the local prediction of the flow (both velocity and temperature fields) along the cold leg and the downcomer, as well as the associated flow-to-wall heat transfer.

Depending on the primary leak size, its location, and the operating conditions, a two-phase configuration may exist, actually resulting from an ECC injection into a partially uncovered cold leg. In such a configuration, Direct Contact Condensation (DCC) and mixing are prime importance phenomena strongly influenced by the interfacial structure and turbulence.

As a consequence, all the interfacial transfers have to be finely accounted for by the required two-phase CFD simulations, both for (i) the highly turbulent bubbly flow in the ECC impinging jet area and (ii) the thermal stratification of the water-steam flow downstream of the ECC nozzle, as well as (iii) the single-phase flow within the downcomer.

The physical validation of the related two-phase CFD codes is a very important issue, and the on-going *TOPFLOW-PTS Experimental Program* was designed in 2005 in this prospect. Its objective is to provide a well documented local data base to further:

(i) complement the validation of the whole CFD modelling in such two-phase configurations;

(ii) improve the understanding of the involved key physical phenomena, and as far as possible enable the validation and/or development of the related models and closure laws.

This paper provides a general overview of the *TOPFLOW-PTS Experimental Program* which is jointly operated and/or financially supported by AREVA (France), Commissariat à l'Energie Atomique (CEA, France), Electricité de France (EDF, France), Swiss Federal Institute of Technology of Zürich (ETHZ, Switzerland), Helmholtz-Zentrum Dresden-Rossendorf (HZDR - formerly FZD, Germany), Institut de Radioprotection et de Sûreté Nucléaire (IRSN, France) and Paul Scherrer Institut (PSI, Switzerland). It successively (i) supports the additional validation data needs, (ii) presents the related experimental setup and some information on the experimental procedure, and (iii) provides the general guidelines of the test matrix definition; eventually, it recaps the main features of the program, and gives some information about the current status of the test performance. It can be considered as an introductory background paper for *TOPFLOW-PTS* companion papers [1] [2] devoted to pre-test CFD calculations, and [3] [4] related to so-called *air-water* (but actually nitrogen-water) test results (see section 4).

2. Identification and specification of additional validation data needs

Additional validation data needs for the CFD modelling of PTS T/H concerns in the two-phase configuration were identified in 2004 within the frame of French joint NEPTUNE Project [5]. They resulted from the application of a generic work approach inferred from a classical validation methodology for (a priori) any industrial application oriented T/H code [6]; actually, they were consistent with those provided a little bit later on during the former stage of the NURESIM project [7].

The main features of this validation methodology and associated work approach, as well as their application results to PTS two-phase configuration were presented in details in [6]. They are briefly reminded.

2.1 Physical validation methodology and associated work approach

Generally speaking, the physical validation of an industrial application oriented T/H code is carried on according to a classical process characterized by three main features: (i) a validation specially relevant for the selected industrial application(s), (ii) a two-step validation approach aiming at the assessment of the dominant basic models against separate-effect test data and of the whole code capability against integral-type test data, and (iii) the use of actually relevant experimental data with respect to the validation aims.

On the basis of this general validation process, a four-step work approach was adopted within NEPTUNE to draw up dedicated validation plans and provide the required relevant validation data, possibly thanks to new experimental programs to be set up. It involved:

(i) a thorough analysis of the concerned industrial applications, to identify the relevant modeling scale, the key physical phenomena and the associated dominant basic models;
(ii) an assessment of these dominant models against the information available for validation, to further specify the additional validation needs and define dedicated validation plans;
(iii) an inventory and assessment of existing validation data to identify and specify the actual needs in new validation data;
(iv) the specification of the new experimental program(s) to provide the additional needed data.

This approach was applied to CFD modeling of the two-phase PTS configuration; its insights are presented in the following section.

2.2 Identification and specification of validation data

The two-phase configuration resulting from a water injection through a water steam flow into a partially uncovered cold leg, involves (see Figure 1):

(i) a dispersed two-phase flow in the ECC impinging jet area. It is mainly a "steam-liquid" bubbly flow, but water drops from the jet might also exist;
(ii) a downstream stratified two-phase flow along the cold leg, and possibly in the downcomer.

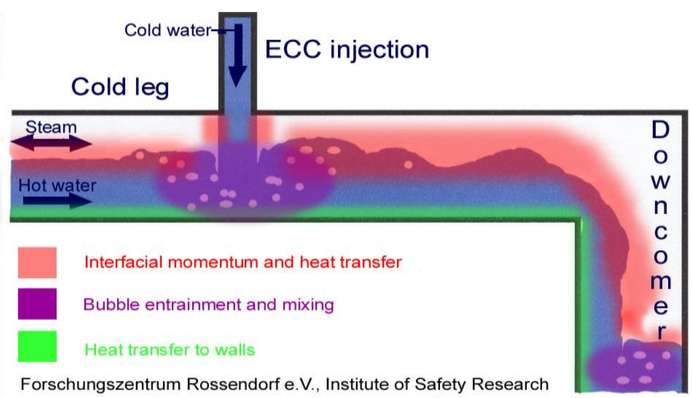

Figure 1 Illustrative scheme of the ECC injection induced configuration

Direct contact condensation and mixing are prime importance phenomena. They are strongly influenced by the interfacial structure and turbulence, and the involved key phenomena are:

(i) DCC along the ECC jet, entrainment, break up and condensation of steam bubbles in the impinging jet area, as well as the turbulence source term. Note this zone is a crucial one since:

- a large part of steam condensation amount takes place in this zone according to CEA COSI experiment results [8],
- it actually defines the boundary conditions for the downstream stratified zone;

(ii) DCC in stratified wavy flows and the turbulent diffusion in presence of density gradients, in the cold leg stratified area;

(iii) turbulence in the downcomer area, with plume effects.

Currently available two-phase CFD codes were not able to accurately simulate all these phenomena (see for instance ECORA synthesis [9]), and the assessment of the associated key basic models had pointed out lacks of information and validation support. A dedicated two-fold validation plan (as afore-introduced) was therefore defined:

(i) it highlighted that the related physical phenomena are mostly very tightly coupled and DCC is important in the whole process, and recommended a further investigation with respect to the actual physical phenomena involved in the real configuration to firm up our knowledge about the nature of the jet and cold leg flows;

(ii) it provided the comprehensive data set needs to correctly carry such a validation plan, i.e:
- liquid and steam phase velocities and temperatures, void fraction and interfacial area, in the impinging jet zone,
- liquid (and possibly steam phase) velocities and temperatures, refined interfacial structure and condensation flux density close to the interface, for the stratified flow zone with density gradients (cold leg and possibly downcomer areas),
- liquid velocities and temperatures in the downcomer, as well as wall temperatures.

The need for the relevance of the related experimental configurations with respect to the industrial application(s) was emphasized.

Several experiments had investigated the key phenomena specifically involved in the two-phase configuration of interest:

(i) either by means of separate-effect tests:
- McKeogh *et al.* [10], Bonetto *et al.* [11], Mansour [12], and Iguchi *et al.* [13] for the impinging jet area,
- Fabre *et al.* [14] for the turbulence in a stratified flow,
- Lim *et al.* [15], Tenschert [16], Noel [17] for DCC,

(ii) or via integral tests (UPTF tests [18]),

but they all present major drawbacks with respect to the afore-defined validation requirements.

Eventually, considering the need for a well-instrumented experiment of the combined phenomena over the whole domain of interest for validation and demonstration purposes with respect to the actual industrial application, it was decided to set up an integral-type experimentation heavily instrumented with state-of-the-art local measuring techniques. The *TOPFLOW-PTS Experimental Program* has been designed in this frame, on the basis of (i) the 900 MWe, CPY plant operated in France, and (ii) a Small-Break LOCA scenario. Initially anticipated within NURESIM project [7], the *TOPFLOW-PTS Experimental Program* is actually being carried on by the above-indicated partner Consortium.

3. Experimental setup

3.1 TOPFLOW facility and so-called “diving tank”

The PTS test section is connected to the HZDR TOPFLOW facility using the so-called “diving chamber technology”:

(i) commissioned in 2002, the TOPFLOW facility [19] is a multi-purpose thermal-hydraulics facility for the investigation of steady-state and transient two-phase flow phenomena in nuclear reactors; it allows performance of two-phase steam-water tests, at up to 286 °C and 7 MPa conditions. It is equipped with a number of standard measurement transducers to control its operation; a special attention has been paid to the reliability and accuracy of the transducers necessary to adjust and provide the test boundary conditions;

(ii) the “*diving chamber technology*” consists in installing the test section in a pressurized tank - so-called *diving tank* - and to operate it in pressure equilibrium with the tank atmosphere. Therefore, the design of the test section is simpler and its manufacturing less expensive, and the thin-walled structures allow the use of specific measurement techniques.

The TOPFLOW *diving tank* is a 7 m long cylindrical capacity, with a 2 m inner diameter. It can be operated at an up to 5 MPa pressure and 50 °C temperature, either with an air or a nitrogen atmosphere.

3.2 Test section geometry

The test section (Figure 2) represents the cold leg with the ECC injection line, a *pump simulator* and a part of the downcomer of the reference CPY plant in a geometrical scale of 1:2.5:

(i) the cold leg geometry is simplified to a straight horizontal pipe; the bend of the reference plant close to the downcomer front wall has been left out to grant a better optical view into the cold leg from the back wall of the downcomer (see section 3.3);

(ii) the flat downcomer model represents a 90° segment of the reference plant one; it gets rid of the original curvature, and its scaled-down length is shortened at both upper and lower ends to fit into the TOPFLOW *diving tank*. A special attention has been paid to:

- the scaling-down of the fluid domain width variations along the downcomer,
- the design of its outlet water box (with the help of CFD calculations) to minimize the flow disturbances caused by the water extraction and as far as possible model a “free” downcomer outlet;

(iii) the *pump simulator* is a vessel located at the upstream end of the cold leg. It represents the coolant volume inside the reference primary pump to model the reference flow conditions upstream from the ECC injection nozzle. It can be supplied with water thanks to a feedwater line at its top, and water-drained at its lowest point.

During test performance, the test section can be heated up to 265 °C whereas the temperature of the *diving tank* atmosphere is upper limited to 50 °C. Therefore the test section has been efficiently insulated, thanks to (i) the direct application of an insulation material on some parts and (ii) an insulated hood set up on other parts requiring an infra-red observation of walls (see section 3.3).

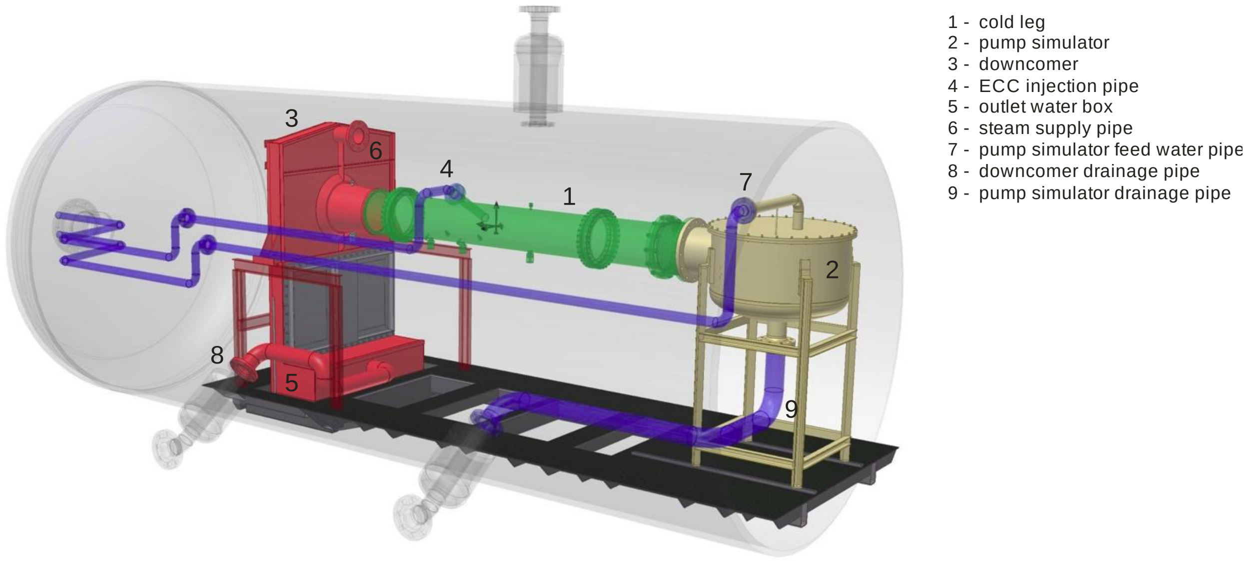

Figure 2 Scheme of the test section located in the diving tank

3.3 Test section instrumentation

Besides the TOFLOW facility standard instrumentation, a dedicated instrumentation is available for the test section related measurements. It comprises in-fluid thermocouples (TC), a "*heat flux probe*" (HFP), wire-mesh sensors (WMS), local void probes equipped with a micro-thermocouple (so-called "*TC-void probes*"), and high-speed and infra-red (IR) camera observations (see Figure 3).

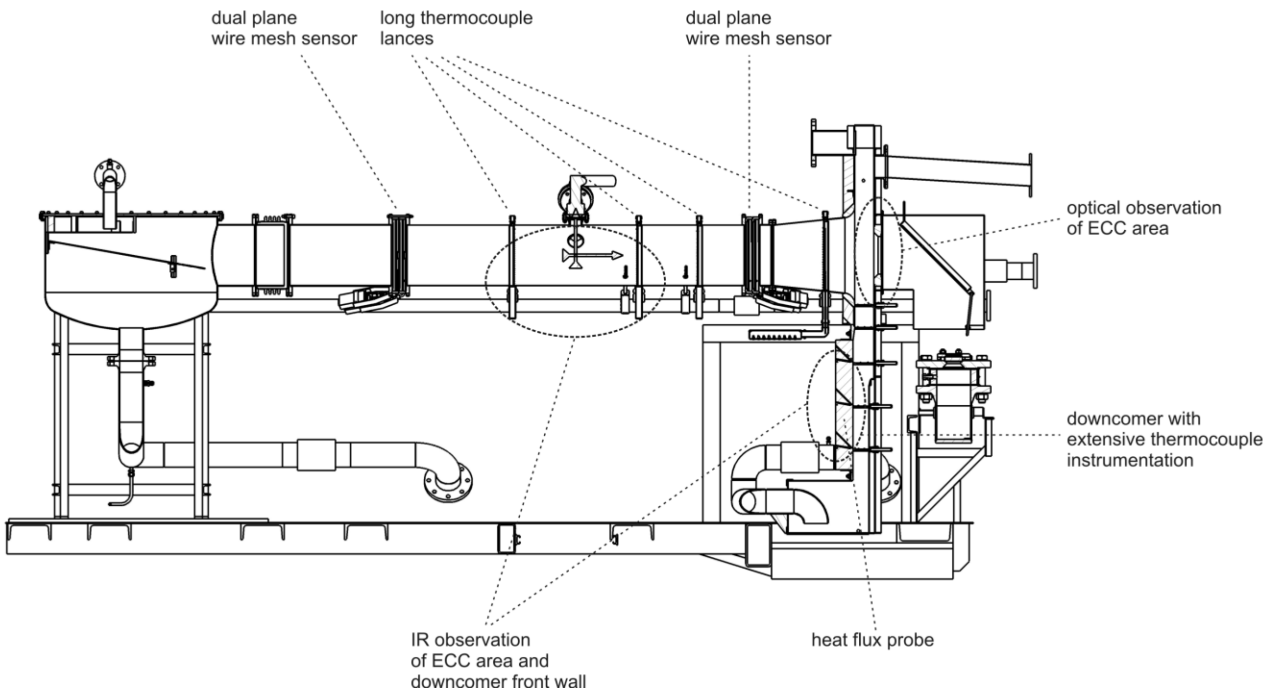

Figure 3 General view of the test section instrumentation

3.3.1 In-fluid TCs

A total number of 169 micro-thermocouples (0.25 mm in sheath outer-diameter, 8 ms in response time) are installed within the flow domain. They are supported with 2 mm thick lances which have a sword-type cross-section thus preventing from any significant effect on the flow field as demonstrated by CFD calculations.

Within the cold leg, there are 112 TCs distributed along:

(i) 4 long lances (with 25 TCs each) mounted vertically along a diameter of the cold leg; three of them (LA1, LA3 and LA4) are located downstream from the ECC injection line orifice, while the fourth is upstream. Their TC distribution has been optimised with respect to the spatial resolution of the expected vertical temperature distributions,

(ii) 4 short lances (with 3 TCs each) implemented in two successive cross-sections to check the 2-D temperature distribution downstream of the injection nozzle.

Additionally, 57 TCs are implemented in the downcomer fluid domain to check the 3-dimensional development of the plume from the cold leg flow discharge in the downcomer to the outlet water box. They are located at 5 axial locations, each with a different number of symmetrical transverse locations (with respect to cold leg axis), thanks to support lances (with 2 or 3 TCs each) fixed on the back wall of the downcomer.

3.3.2 Heat flux probe

In order to assess the capability of the whole CFD modeling during transient tests, including the flow-to-wall heat transfer, a *heat flux probe* has been designed with the help of coupled "CFD-heat conduction" pre-test simulations. It consists of a 600 mm high x 200 mm wide x 80 mm thick steel bar instrumented with a 3x3x3 matrix of thermocouples and integrated in the front wall of the downcomer model; it is positioned in such a way that its 2-D (axial and transverse) TC distribution corresponds to the in-downcomer TC one.

3.3.3 Wire-mesh sensors (Figure 4)

On the one hand, two pairs of two-layer WMSs, with a 32x32 measuring point matrix each, are upright located in two cross-sections of the cold leg, respectively upstream from the ECC injection nozzle and close to the downcomer inlet nozzle. On the other hand, a three-layer WMS with 2x64x64 measuring points can be placed into the impinging jet area in an inclined position perpendicularly facing the ECC injection nozzle.

These sensors were developed at HZDR. They will supply high-resolution information on (i) bubbly gas phase (void fraction profiles, flow structure including interfacial area concentrations, bubble-size distributions, gas phase velocity) - if any, and (ii) water level and surface wave structures, along the cold leg; they also might provide liquid velocity thanks to cross-correlation techniques presently under development at ETH Zürich within *TOPFLOW-PTS Experimental Program*.

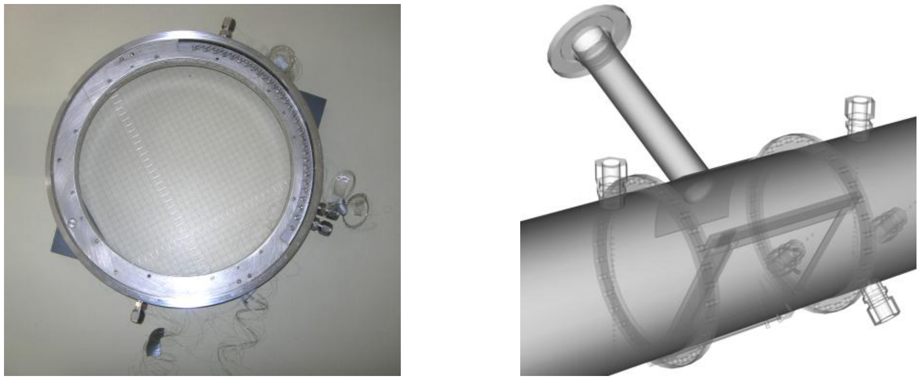

Figure 4 Upright and inclined wire-mesh sensors for cold leg and jet zone investigation

3.3.4 TC-void probes

TC-void probes combine fast phase detection with fast local temperature measurement. Six such probes will be arranged in the impinging jet area opposite to the injection orifice, and operated in combination with the large inclined wire-mesh sensor.

3.3.5 High-speed and infra-red camera observations

Flow observation in the impinging jet zone is made thanks to:

(i) a fiber optic illumination through a glass window at the top of the cold leg, right above the injection region,

(ii) an optical viewport located at the back wall of the downcomer in front of the cold leg, and an optical mirror to direct the view into the cold leg pipe,

(iii) a “Motion Pro” high speed camera (500 frames per second for a maximal 1280x1024 pixel resolution).

This set-up will mostly provide qualitative insights; its successful operation has been demonstrated with a dedicated experiment on a cold leg model.

An infra-red camera with a 50 frames-per-second speed and a 320x240 pixel resolution, and IR mirrors are used for the thermography of the bottom and side parts of the cold leg wall in the impinging jet area, or of the front wall of the downcomer. Considering the low thermal inertia of the thin walls, it will provide a valuable estimate of the “wet temperature” at the wall inner-surface, mainly during transient tests.

Each camera needs to be located in the *diving tank* and is prior-enclosed into a protective, pressure-proof container.

Note the afore-presented possible measurements cannot be made all together during one single test, since there are experimental antagonisms for their performance at the same time Therefore, different actual instrumentation configurations are defined and set up along with the different test series.

4. Guidelines for the test matrix definition

The test matrix definition is based on the following guidelines:

(i) performance of steady-state and transient tests, with and without mass transfer i.e. steam-water and *air-water* tests respectively, to check the importance of phenomena coupling;

(ii) variation of T/H test conditions (ECC flowrate and ECC-to-cold leg temperature difference, and pressure, as well as water level in the cold leg) with respect to reference scaled-down conditions to tentatively investigate key PTS phenomena, mostly during the steady-state tests.

The reference conditions for TOPFLOW-PTS tests are based on a Froude scaling of the reference NPP conditions, further considering the 5 MPa pressure upper limit for the tests; the reference NPP conditions result from a CATHARE simulation of the selected SB-LOCA scenario for the French 900 MWe-CPY plant.

The scaling is based on a Froude similarity related to either the momentum free surface induced-effects in the impinging jet area, or the mixing/stratification processes in the liquid flow of the cold leg under the gas-liquid interface. Associated Froude numbers to be considered respectively stand as:

$$Fr_{LG} = \sqrt{\frac{\rho_{ECC} \cdot V_{ECC}^{2}}{(\rho_{ECC} - \rho_{G,CL}) \cdot g \cdot L}}$$

$$Fr_{LL} = \sqrt{\frac{\rho_{ECC} \cdot V_{ECC}^{2}}{(\rho_{ECC} - \rho_{L,CL}) \cdot g \cdot L}}$$

where:

- ρ_{ECC}, $\rho_{L,CL}$ and $\rho_{g,CL}$ are the density of the ECC water, liquid and gas phases in the cold leg, respectively,
- V_{ECC} is the averaged velocity of the ECC water at the injection nozzle,
- L is a reference dimension,
- g is the gravitational acceleration.

Note that:

(i) the Froude similarity for the Kelvin-Helmholtz instability of the gas-liquid interface along the cold leg cannot be adjusted during the tests since the gas velocity is not controlled. Its impact can be however investigated (to a certain extend) via pressure variation;

(ii) the Weber similarity may play a role in the impinging jet area (jet fragmentation, bubble entrainment, coalescence and break up). Temperature variations during the air-water tests should help in estimating its effect;

(iii) a special attention has to be paid to the existence of a gas-water flow separation in the ECC injection line at a flowrate threshold, as pointed out by pre-test calculations [1] [2]. Dedicated tests are performed with respect to this issue.

5. Experimental procedure

Tests are performed with a nitrogen atmosphere to prevent from any possible combustion of materials inside the *diving tank* where test pressure is controlled, whereas a full permanent pressure equilibrium operation of the test section is ensured thanks a "test section – *diving tank*" connection via a condenser unit (see Figure 5). For steam-water tests, steam is provided to the test section at the top of the downcomer and the extra steam amount flows through the condenser unit where it condenses; for *air-water* tests, the test section is provided with nitrogen from the diving tank inventory directly via its connection to the diving tank (in this case, the condenser unit has a passive role).

Steady-state tests are performed with a water supply and possible drainage of the *pump simulator*, to actually get a thermal stratification all along the cold leg; to minimise its induced momentum effect, the water supply flows on a slightly inclined distribution plate. Transient test are performed with no water supply or drainage of the *pump simulator*, and no distribution plate.

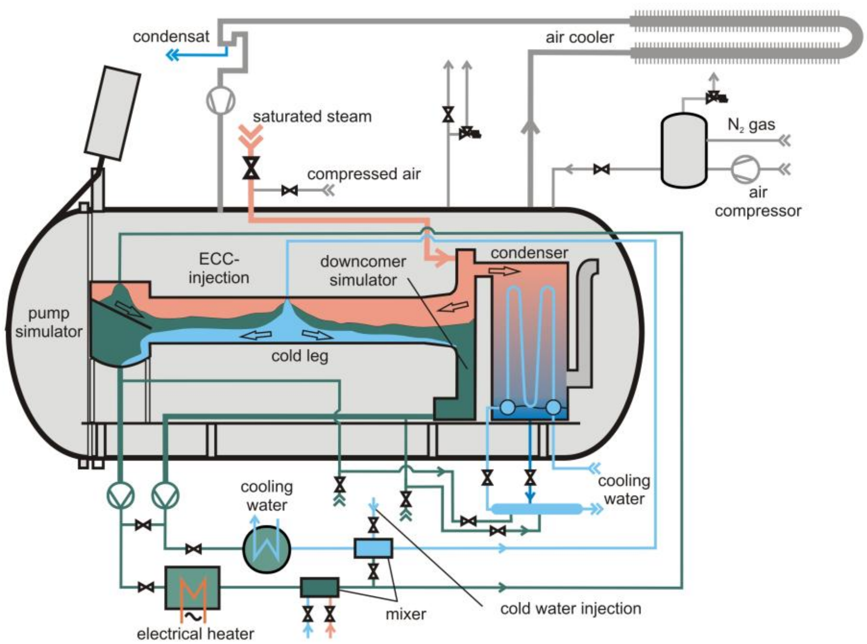

Figure 5 Scheme of experimental setup

6. Conclusion. Prospects

Jointly operated and/or supported by AREVA, CEA, EDF, ETHZ, HZDR, IRSN and PSI, the *TOPFLOW-PTS Experimental Program* is an integral-type program devoted to the CFD modelling of a PTS two-phase flow configuration. It aims at both complementing the validation of the whole modelling for a typical industrial configuration, and improving the understanding and modelling of the involved key physical phenomena. In this frame:

(i) it as far as possible accounts (considering the experimental constraints) for a correct relevance of the experimental configuration with respect to the industrial application one, on both geometrical and thermal-hydraulics points of view;

(ii) it involves a dense implementation of different state-of-the art measurement techniques in the impinging jet area, along the cold leg and downcomer, to provide local data sets for the related T/H fields;

(i) it includes performance of transient and/or steady-state tests, with and without mass transfer (i.e. with water ECC injection in a steam and a nitrogen environment, respectively), up to a 5 MPa pressure.

Pre-test CFD calculations have been performed and have been a help for both the experimental setup design and the test matrix definition.

Some *air-water* test series dealing with (i) the flowrate threshold resulting in a gas-liquid flow separation in the ECC injection line and (ii) the thermal stratification along the cold leg and the downcomer, were performed in late 2009 and early 2010. Some main results are presented in companion papers [3] [4]. Steam-water tests are being carried on.

7. References

[1] A. Martin *et al.*, "Pre-test calculations of TOPFLOW-PTS experiment with NEPTUNE_CFD code", Proceedings of NURETH-14 Conference (paper n° 350), Toronto, Ontario, Canada, September 25-29, 2011.

[2] P. Apanasevich *et al.*, "Numerical simulations of the TOPFLOW-PTS steam-water experiment", Proceedings of NURETH-14 Conference (paper n° 362), Toronto, Ontario, Canada, September 25-29, 2011.

[3] B. Niceno *et al.*, "Reynolds Averaged Navier-Stokes (RANS) and Large-Eddy Simulations (LES) of the air-water TOPFLOW-PTS experiment", Proceedings of NURETH-14 Conference (paper n° 608), Toronto, Ontario, Canada, September 25-29, 2011.

[4] T. Seidel *et al.*, "TOPFLOW-PTS air-water experiments on the stratification in the ECC nozzle and the ECC water mixing during PTS scenarios", Proceedings of NURETH-14 Conference (paper n° 394), Toronto, Ontario, Canada, September 25-29, 2011.

[5] A. Guelfi *et al.*, "A new multi-scale platform for advanced nuclear thermal-hydraulics. Status and prospect of the NEPTUNE project", Proceedings of NURETH-11 Conference, Avignon, France, 2005 October 2-6.

[6] P. Péturaud and E. Hervieu, "Physical validation strategy of the NEPTUNE two-phase modelling: validation plan to be adopted, experimental programs to be set up and associated instrumentation techniques developed, Proceedings of NURETH-11 Conference, Avignon, France, 2005 October 2-6.

[7] D. Cacuci *et al.*, "NURESIM: a European Platform for Nuclear Reactor Simulation", Proceedings of the Conference on EU Research and Training in Reactor Systems (FISA 2006), Luxembourg, 2006 March 10-13.

[8] A. Janicot and D. Bestion, "Condensation modeling for ECC injection", *Nuclear Engineering & Design*, Vol. 145, 1993, pp. 37-45

[9] M. Scheuerer *et al.*, "Evaluation of Computational Fluid Dynamics for Reactor Safety Analysis. Condensed Final Summary Report", EC ECORA Project n° FIKS-CT-2001-00154, 2001.

[10 E.J. McKeogh and D.A. Ervine, "Air entrainment rate and diffusion pattern of plunging liquid jets", *Chem. Engineering Science*, Vol. 36, 1980, pp. 1161-1172.

[11] F. Bonetto and R.T. Lahey, "An experimental study on air carryunder due to a plunging liquid jet", *Int. J. Multiphase Flow*, Vol. 19, No. 2, 1993, pp. 281-294.

[12] A. Mansour and N. Chigier, "Turbulence characteristics in cylindrical liquid jets", *Phys. Fluids*, Vol. 6, No. 10, 1994, pp. 3380-3391.

[13] M. Iguchi *et al.*, "Mean velocity and turbulence characteristics of water flow in the bubble dispersion region induced by plunging water jet", *Int. J. Multiphase Flow*, Vol. 24, No. 4, 1998, pp. 523-537.

[14] J. Fabre *et al.*, "Stratified flow. Part I : local structure, *Multiphase Science and Technology*, Vol. 3, 1987, Ed. by G. F. Hewitt, J. M. Delhaye, and N. Zuber, pp. 285-301.

[15] I.S. Lim *et al.*, "Condensation measurement of horizontal cocurrent steam-water flow", *Journal of Heat Transfer*, 106,1984, pp. 425-432.

[16] J. Tenschert, "HDR test series E33.1/2 for investigation of contact condensation processes in horizontal pipes during emergency core cooling injection", *Nuclear Engineering and Design*, Vol. 154, 1995, pp. 1-16.

[17] B. Noël, "Synthèse des enseignements tirés des essais COSI", Internal CEA report 2001, SETEX/LETS/2001-145.

[18] P.A. Weiss and R.J. Hertlein, "UPTF Tests Results. First 3 Separate-Effect Tests", 14th Water Reactor Safety Research Information Meeting, Gaithersburg, Maryland, USA, 1986.

[19] A. Schaffrath et al., "TOPFLOW – a new multipurpose thermalhydraulic test facility for the investigation of steady state and transient two phase flow phenomena", *Kerntechnik*, Vol. 66, 2001, pp. 209-213.

ACKNOWLEDGEMENTS

The authors are grateful to the HZDR TOPFLOW facility team for their provision of engineering drawings and their contribution to the project.