The 14th International Topical Meeting on Nuclear Reactor Thermal Hydraulics, NURETH-14
Toronto, Ontario, Canada, September 25-30, 2011

NURETH14-350

TOPFLOW-PTS EXPERIMENTS. PRE-TEST CALCULATIONS WITH NEPTUNE_CFD CODE

A. Martin*[1], C. Heib[2], F. Dubois[2], C. Raynaud[1], P. Péturaud[1], F.Huvelin[3], A. Barbier[3]
[1] Electricité de France Chatou, France
[2] Institut de Radioprotection et de Sûreté Nucléaire (IRSN), Fontenay-aux-Roses, France
[3] AREVA-NP, Paris la Défense, France
alain-cc.martin@edf.fr, caroline.heib@irsn.fr, franck.dubois@irsn.fr, christelle.raynaud@edf.fr, pierre.peturaud@edf.fr, fabien.huvelin@areva.com, anthony.barbier@areva.com

Abstract

Hypothetical Small Break Loss Of Coolant Accident is identified as one of the most severe transients leading to a potential huge Pressurized Thermal Shock on the Reactor Pressure Vessel (RPV). This may result in two-phase flow configurations in the cold legs, according to the operating conditions, and to reliably assess the RPV wall integrity, advanced two-phase flow simulations are required. Related needs in development and/or validation of these advanced models are important, and the on-going TOPFLOW-PTS experimental program was designed to provide a well documented data base to meet these needs. This paper focuses on pre-test NEPTUNE_CFD simulations of TOPFLOW-PTS experiments; these simulations were performed to (i) help in the definition of the test matrix and test procedure, and (ii) check the presence of the different key physical phenomena at the mock-up scale.

1. Introduction

In the frame of lifetime extension of Pressurized Water Reactors (PWR), Pressurized Thermal Shock (PTS) along the Reactor Pressure Vessel (RPV) wall is one of the most crucial concerns with respect to its integrity. To assess the RPV integrity and evaluate the lifetime of French (PWR type) Nuclear Power Plants (NPP), EDF and AREVA have developed a three-step methodology including (Figure 1) (i) system-code calculations to define the transients related scenarios and associated boundary conditions needed for the further step, (ii) coupled "CFD-Heat conduction" simulations to determine the thermal loads within the RPV walls, and (iii) a subsequent mechanical analysis.

Related analyses have shown that one of the most severe loading conditions is given by the Small Break Loss Of Coolant Accidents (SBLOCA). During this type of hypothetical LOCA, cold Emergency Core Cooling (ECC) water is injected into the cold leg and partially mixes with the hot primary fluid flow down to the downcomer, possibly resulting in a large PTS on mechanical structures, first of all on the RPV wall. Depending on the accident transient scenario - leak size and location for example - and on the operating conditions of the considered NPP, a single-phase or two-phase flow configuration can exist in the cold legs.

Actually, the current RPV wall integrity demonstration has to consider accident transients resulting in two-phase flow configurations, and further requires dedicated two-phase flow CFD simulations. Direct Contact Condensation (DCC), mixing and flow-to-wall heat transfer are of prime importance in this situation. These phenomena are strongly influenced by the structure of the water-steam interface and by turbulence, and related interfacial transfers (mass, momentum – including turbulence – and energy) have then to be reliably represented by the CFD simulations. It is thus necessary to develop and/or validate specific models, which have to accurately represent (i) the highly turbulent bubbly flow in the ECC impinging jet area, and (ii) the thermal stratification of the water-steam flow along the cold leg downstream from the ECC nozzle, as well as (iii) the single-phase flow in the down comer, including the flow-to-wall heat transfer along the cold leg and RPV wall.

After a first R& D work stage being performed for several years, a significant extensive engineering work devoted to the RPV structural integrity re-assessment for the French 900 and 1300 MWe PWRs is in progress at EDF and its French partners ([1] to [3]). Numerous developments have been implemented or are in progress to make sure the CFD methodology reliably represents the flow during a PTS event. The on-going TOPFLOW-PTS experimental program [4] was designed in this frame. It aims at providing a well documented database to further (i) complement the validation of the whole CFD modelling in PTS two-phase configurations, (ii) improve the understanding of the involved key physical T/H phenomena, and (iii) as far as possible enable the validation and/or development of the related models and closure laws.

After a general overview of TOPFLOW-PTS experimental program, this paper is devoted to pre-test CFD simulations of TOPFLOW-PTS experiments including both air/water and steam/water configurations. Such pre-test simulations were carried out with NEPTUNE_CFD code to help in the definition of the test matrix, and possibly of the test procedure, checking to what extent the different key physical phenomena of interest are present and can be "caught" by the implemented instrumentation.

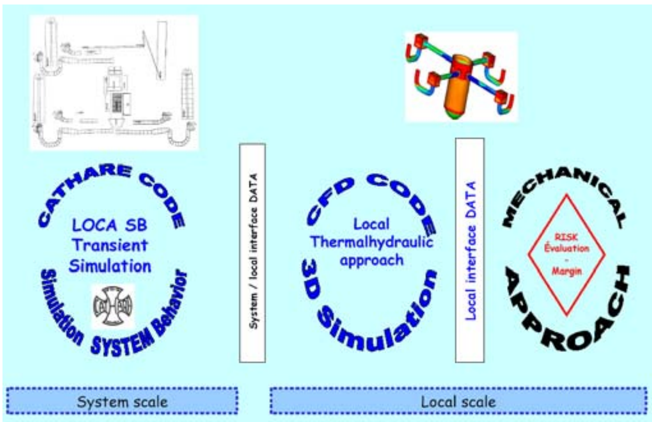


Figure 1 Chaining computation methodology

2. TOPFLOW-PTS experimental setup

2.1 General

The TOPFLOW-PTS was designed in 2005 after the identification (within the frame of French joint NEPTUNE Project [5]) of pending validation data needs for the CFD modeling of PTS Thermal Hydraulics (T/H) concerns in the two-phase configuration. Actually, these validation data needs were related to the local modelling of the afore-mentioned key physical phenomena and a priori required separate-effect experiments; however, considering:

(i) these phenomena are mostly very tightly coupled and DCC is important in the whole process, and

(ii) 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 – TOPFLOW-PTS - heavily instrumented with state-of-the-art local measurement techniques.

The TOPFLOW-PTS experimental program is based on (i) the 900 MWe CPY NPPs operated in France, and (ii) a SB LOCA scenario. It 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).

2.2 Main features

The PTS test section (Figure 2) models the cold leg with the ECC injection line, a primary pump and a part of the downcomer of the reference CPY plant in a 1:2.5 geometrical scale, with some simplifications i.e. (i) a straight cold leg, (ii) a flat downcomer representing a 90° segment of the reference plant, and (iii) a pump simulator – actually a vessel located upstream from the ECC injection nozzle. This pump simulator can be supplied with water at its top and water-drained at its lowest point, so that steady-state tests can be achieved with an actual thermal stratification along the cold leg.

This test section is connected to the HZDR TOPFLOW facility and installed in the so-called *diving tank*, which allows test performance at up to 5 MPa and 265 °C conditions, at pressure equilibrium with the tank nitrogen atmosphere, along with a thin-walled test section design for implementation of infra-red thermography. Its is instrumented with:

(i) 196 thermocouples distributed within the cold leg and downcomer flow, as well as in a so-called *Heat Flux Probe* – HFP (it is actually a thick steel bar integrated in the downcomer front wall);

(ii) wire mesh sensors along the cold leg (impinging jet and stratified flow areas), as well as "*TC-void probes*" (impinging jet area), to provide information on the bubbly gas phase, water level and surface wave structure, and possibly liquid phase velocity;

(iii) a high-speed camera for the observation of the impinging jet area from the back wall of the downcomer, and an infra-red camera for a thermography of the cold leg wall and of the downcomer front wall.

The test program includes (on-going) performance of transient and/or steady-state tests, with and without mass transfer (i.e. with water ECC injection in a steam or a nitrogen environment, respectively), up to a 5 MPa pressure. Test parameters are varied around the scaled-down reference condition set.

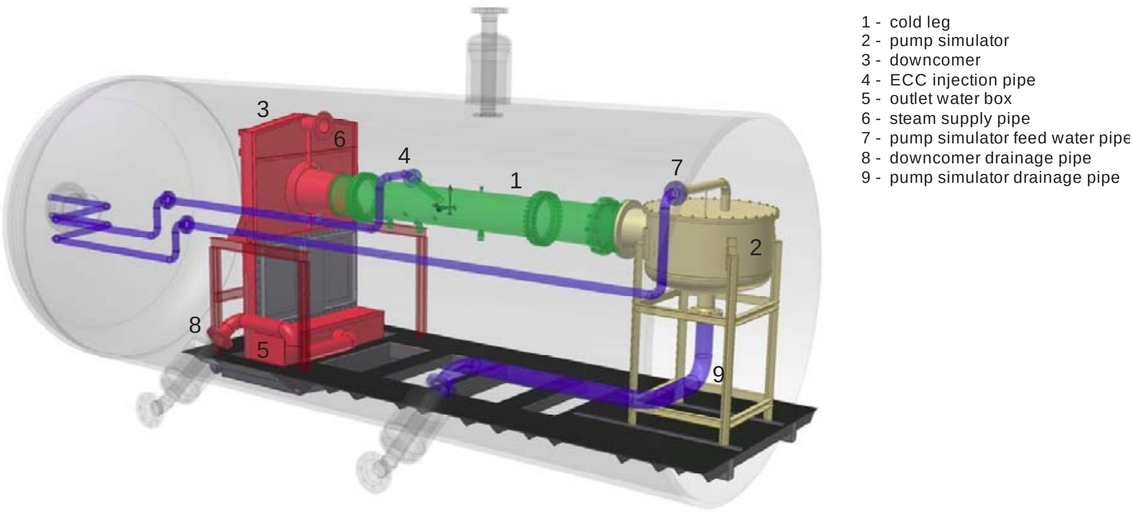


Figure 2 Test section scheme

3. NEPTUNE_CFD computer code

3.1 General features

NEPTUNE_CFD is the multi-phase CFD solver of the NEPTUNE software platform [5], jointly developed by CEA and EDF with a complementary support of AREVA and IRSN. This platform is dedicated to advanced two-phase flow thermal-hydraulics, allowing multi-scale and multi-disciplinary calculations meeting the industrial needs (among which PTS in two-phase configuration). NEPTUNE_CFD has also been chosen as a basis for the development of CFD tools in the international shared software platform for nuclear thermal-hydraulics developed in the European framework, i.e. former NURESIM [6] project and further today NURISP project (see [7] presented in CFD4NRS conference).

The basic model of NEPTUNE_CFD is the classical six-equation two-fluid model (mass, momentum and energy for both liquid and gas, with the same pressure for the two phases), coupled to a k-ε model for the continuous phases, and (optionally) a transport equation for the interfacial area concentration.

From a numerical point of view, the spatial discretization is based on a co-located finite volume approach that accepts meshes with any type of cell (tetrahedral, hexahedral, prismatic, pyramidal, polyhedral…) and any type of grid structure (unstructured, block structured, hybrid, conforming or with hanging nodes…), while the solver algorithm is an original method dedicated to multi-phase systems [8].

3.2 PTS-related model developments in NEPTUNE_CFD

In the frame of NEPTUNE_CFD development for PTS-related application, a special attention is paid to the modelling of (i) the detection of the position of the gas-liquid interface in the stratified flow area, (ii) the turbulence, especially close to the free surface, and (iii) the heat and mass transfer close to the free surface at the gas-liquid interface and in the impinging jet area.

4. NEPTUNE_CFD pre-test simulations

4.1 Objectives

Initially, the NEPTUNE_CFD pre-test simulations aimed at refining the test section design and helping the definition of the test matrix and experimental procedure; in this frame, the related result analyses checked to what extend the key physical phenomena of the industrial scenario are present at the mock-up scale and with the foreseen operating conditions.

4.2 Geometry and grid

Figure 3 illustrates the CAD model of the TOPFLOW-PTS test section.

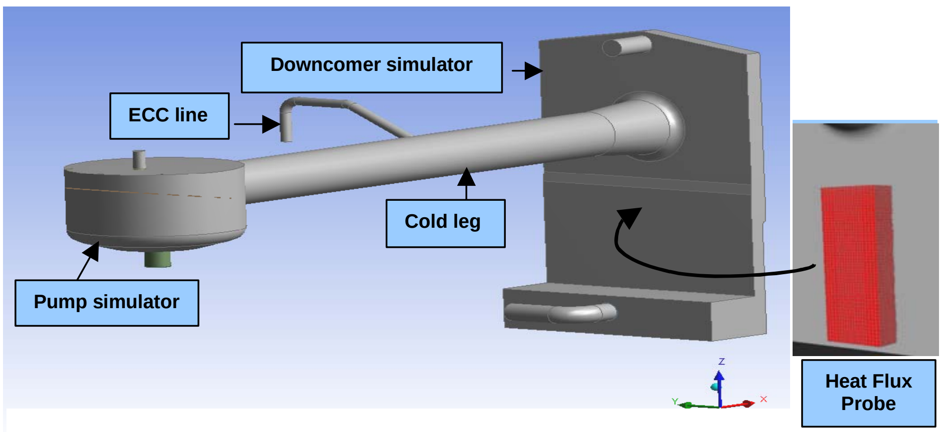


Figure 3 CFD model of the TOPFLOW-PTS test section

The fluid domain is represented with 431 330 hexagonal cells whereas the HFP solid mesh is represented with 58320 tetrahedral cells.

4.3 Initial and boundary conditions

The allowed boundary conditions in terms of mass flow rate and temperature are represented in Figure 4. There are:

(i) 3 fluid inlets: the cold water inlet at the ECC injection nozzle, the "hot" feedwater inlet at the top of the pump simulator and the gas inlet at the top of the downcomer. Whereas the temperature and mass flow rate are explicitly imposed for the ECC injection and pump feed water, they step-by-step result from the simulation for the gas inlet at the top of the downcomer;

(ii) 3 fluid outlets: the water outlet at the bottom of the downcomer, the gas outlet at the top of the downcomer and the water outlet at the bottom of the pump simulator (to drain the flow coming from the cold leg). Note only the latter outlet flowrate is prescribed as an explicit boundary condition; the two former ones are based on simulation results at each time step to respectively maintain a constant water level in the cold leg and a constant pressure in the test section atmosphere.

Finally, at the beginning of the calculation, the cold leg is (otherwise differently specified) filled in water and the water level is decreased (with the ECC injection in operation) down to the cold leg mid-plane ("*50 % level*") during the first step of the simulation; during the subsequent second step, it is maintained at a 50 % value. Section 6.2 devoted to the gas-liquid flow separation occurrence in the ECC line will point out the importance of this procedure.

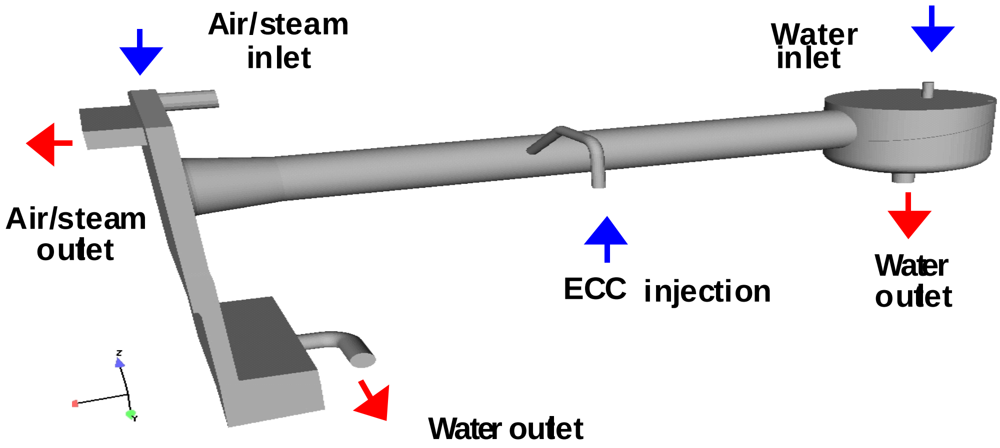


Figure 4 Boundary conditions allowed for the TOPFLOW-PTS simulations

4.4 Physical properties, physical and numerical models

For water and steam, the CATHARE water/steam properties (implemented in the NEPTUNE_CFD code) were used to calculate density, thermal diffusivity, specific heat and dynamic viscosity over a large range of temperature and pressure. For air, the needed properties were assumed as constants.

For the solid domain, specific heat and thermal conductivity were dependent on temperature, but not density.

The Large Interface Model [9] was used for the modeling of the free surface. Heat transfers between phases were modeled only for the steam/water cases. Simulations were performed with a thermal coupling between the fluid and the solid domains, with a time step computed in order to respect a CFL lower than 1.

5 Results and analysis

5.1 Pre-test simulation series performed and associated objectives

Two pre-test simulation series were performed. They were respectively focused on:

(i) the flow pattern in the ECC injection line, with the possible existence of a gas-liquid flow separation. Parametric simulations in air/water and steam/water configurations were thus carried out in turn, to check the related ECC flowrate threshold, the existence of any possible hysteresis phenomenon as well as of any test conditions and procedure impact;

(ii) the presence of T/H key phenomena in the foreseen experimental tests; one air/water and one steam/water configurations (with constant boundary conditions) were considered. The qualitative and quantitative analyses were mainly related to the thermal stratification and associated gradient amplitude in the cold leg and downcomer, over the transient operation till the steady-state achievement. A peculiar attention was also paid to the thermal gradients in the *Heat Flux Probe*, to check to what extend the related measurement results would be sufficient for the further analyses.

Their results are presented below, along with their insights with respect to the test procedure and test matrix definition. Note that no sensitivity study (related to grid density, physical and numerical models) was performed during this pre-test simulation stage, considering its actual aims; such sensitivity study will be carried on during the post-test simulation stage.

5.2 Investigation of gas-liquid flow separation in ECC injection line

It is very important, for both the test matrix and procedure definition, and further data interpretation, to foresee if, according to prescribed conditions, a gas-liquid flow separation may occur in the ECC injection line. Indeed, such a phenomenon occurrence has a direct impact on the outwards ECC jet form, velocity distribution and impulse, which results in a huge influence on the mixing process in the impinging jet area and downstream cold leg stratified area (including the free surface behaviour).

The related pre-test investigation was carried out thanks to NEPTUNE_CFD simulations in air/water and steam/water configurations, according to a two-step procedure:

(i) firstly, a gradual uncovering of the cold leg from an initial water-filled configuration down to a 50 % water level, along with an ECC injection at a large enough constant flowrate providing a single-phase flow in the injection line;

(ii) then, a continuously time-decrease of ECC flowrate down to the flowrate threshold resulting in a clear gas-liquid flow separation along the ECC line, followed by a flowrate increase to check for any possible hysteresis. The latter increase was either a continuous one or a step-by-step one.

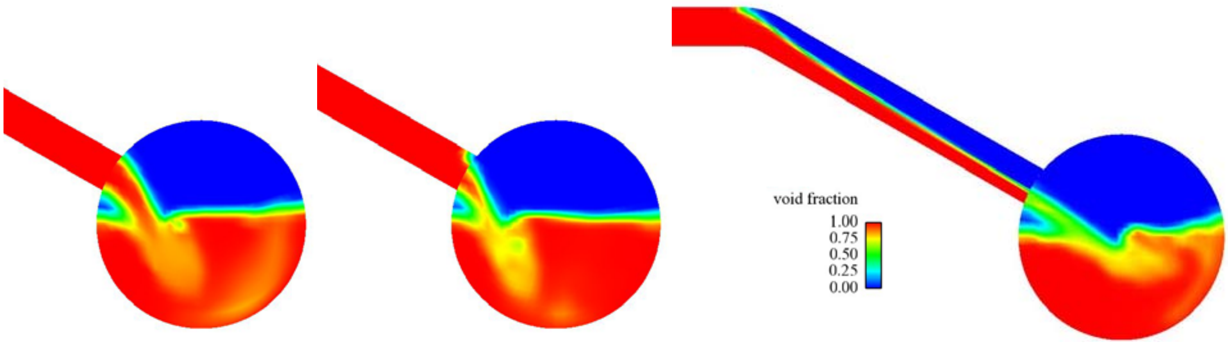
For both air/water and steam/water configurations, this CFD-based investigation showed that during the ECC flowrate decrease stage (Figure 5) (i) a first gas "bubble" appears at the upper circumferential location of the ECC line orifice (onset of flow separation) and (ii) the gas phase upstream propagates along the upper section of the ECC line after a slight additional ECC flowrate decrease. The flowrate threshold providing a clear gas-liquid flow separation in the ECC line is very similar for the two fluid configurations, but it slightly depends on the initial configuration of the cold leg: if the ECC injection starts while the cold leg is already (partially) uncovered instead of being water-filled, the flowrate threshold resulting in the gas-liquid flow separation in the ECC line might be higher than previously-obtained one, depending on initial ECC flowrate.

For the subsequent ECC flowrate increase stage, the investigation demonstrated the existence of a hysteresis phenomenon (with respect to the flow separation in the cold leg), much higher for the air/water configuration than for the steam/water one. Actually with the same ECC flowrate range, water-filled ECC line was not recovered in the former configuration whereas it was in the latter one, with a higher flowrate than the afore-computed threshold; furthermore note that the hysteresis amplitude depends on the characteristics of the flowrate increase evolution: the continuously time-increase evolution has resulted in a higher "recovering flowrate" than the step-by-step one.

These different insights resulted in recommendations for both:

(i) the test matrix definition, which took care of the anticipated ECC flowrate threshold values, and planned dedicated tests with respect to this ECC line flow separation concern;

(ii) the test procedure. Tests are to be performed with an initially water-filled cold leg.



(a) (b) (c)

Figure 5 Flow pattern in the ECC injection line: (a) single-phase flow, (b) gas "bubble" at the ECC line orifice, and (c) gas-liquid flow separation along the ECC line

5.3 Check of the representation of the key physical phenomena

5.3.1 Operation of the pump simulator

As previously indicated (§ 2.2), hot water is supplied at the top of the pump simulator to establish a counter-current flow along the cold leg upstream from the ECC injection orifice (and also to allow a faster steady state achievement), whereas water can be drained at its lowest point to avoid unphysical accumulation of hot water upstream from the ECC line.

A very first pre-test simulation of an air/water configuration indicated that the joint water supply and drainage of the pump simulator resulted in the formation of an undesired circulation loop (Figure 6). The injected hot water does not penetrate much into the cold leg and rather tends to directly flow into the pump simulator to be further drained. The formation of this loop prevents from reaching the foreseen goal, and it was decided that the air/water steady-state experimental tests would be performed without any drainage of the pump simulator.

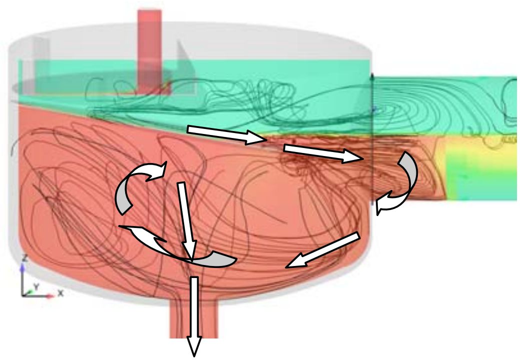


Figure 6 Flow loop in the pump simulator

5.3.2 Investigation of the thermal stratification in the cold leg

(i) Air water configuration (Figure 7)

By buoyancy and dynamic effects, ECC cold water from the injection line flows towards the bottom of the cold leg, initially filled with hot water. After a few seconds, it reaches the cold leg wall on the opposite side of the ECC line orifice, leading to a strong mixing and cooling in the impinging jet area. The cold water flows upstream towards the pump and downstream towards the downcomer while forming a thermal stratification; therefore, a reverse flow of hot water appears in the upper part of the cold leg.

Since there is a hot water injection upstream from the impinging jet area (coming from the water feeding the pump simulator), a thermal stratification is observed in the upstream part of the cold leg. Downstream from the impinging jet area, the thermal stratification weakens with time since there is no hot source to maintain the stratification, and finally, a perfect mixing is obtained. Figure 7 which represents the liquid temperature T evolution in the cold leg illustrates this final state (T_{max} and T_{ECC} are respectively the initial cold leg temperature and ECC temperature).

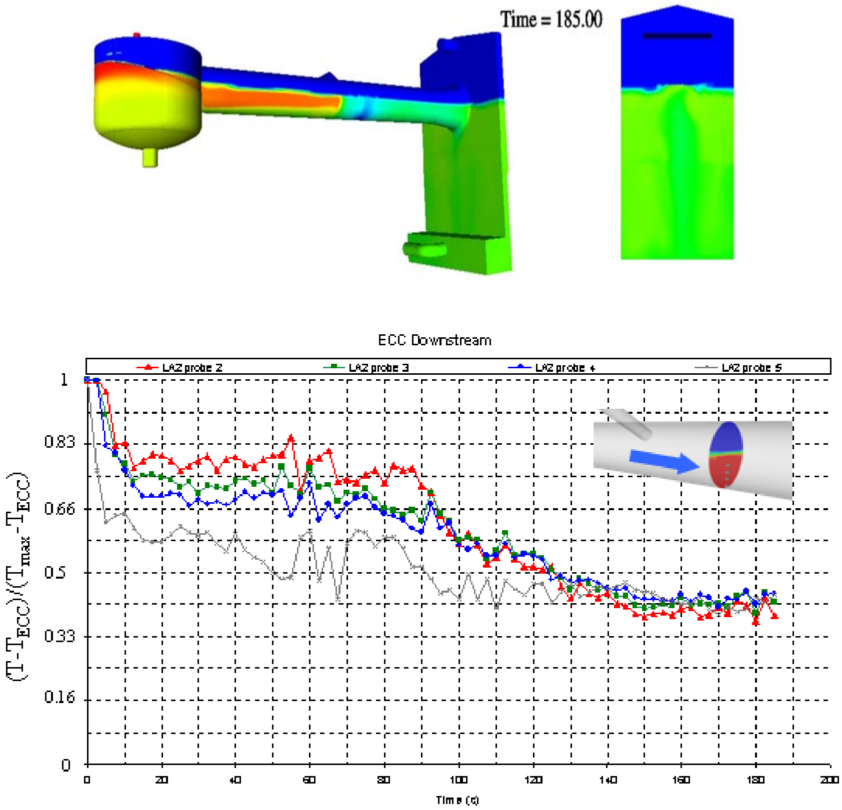


Figure 7 Air/water configuration. Stratification in cold leg

As an important result, only the initial transient stage of air/water configuration tests is interesting and can provide valuable information regarding TOPFLOW-PTS program purposes; there is definitely no relevant use of the final steady-state operation.

(ii) Steam/water configuration (Figure 8)

The same physical phenomena as those described for the air/water configuration are observed in the steam/water configuration.

But direct contact condensation of steam along the free surface allows to maintain a thermal stratification on both sides of the ECC nozzle, and to eventually achieve a final steady-state stratification along the cold leg, with very valuable measurement results for the assessment and validation of the involved CFD models. Figure 8 illustrates this final state with the stratification and its thermal gradient all along the cold leg (T_{sat} is the initial temperature of water and steam in the cold leg).

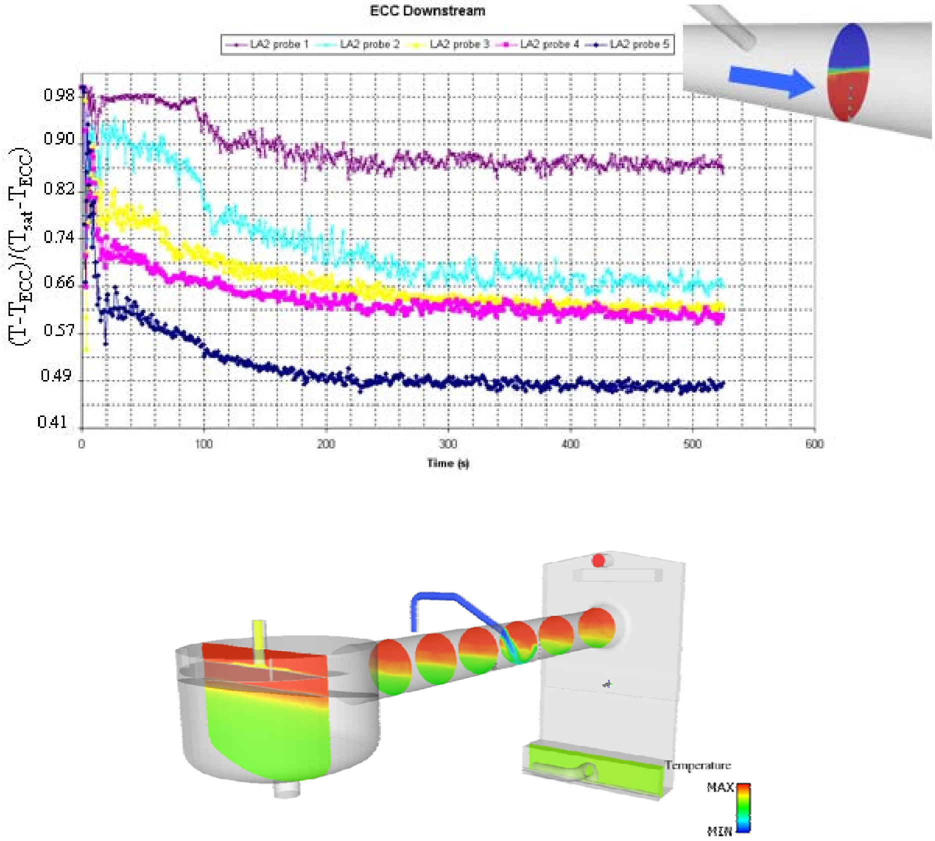


Figure 8 Steam/water configuration. Stratification in cold leg

5.3.3 Investigation of the temperature gradient in the Heat Flux Probe

For the afore-presented steam/water configuration simulation, the time-evolution of the wall temperature distribution in the heat flux probe is plotted at the location of the immersed thermocouple (Figure 9).

It shows that the related thermocouple measurements within the as-designed HFP can result in significant temperature differences along a transient PTS-type test, and will be very useful to a assessment of the whole modeling involved in the coupled "CFD-heat conduction" simulation of a PTS scenario.

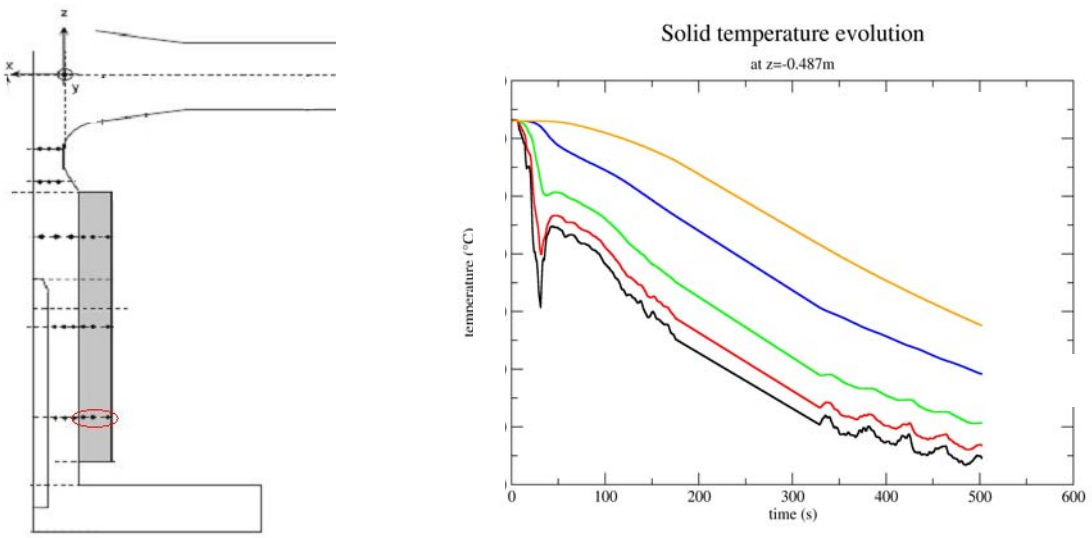


Figure 9 Temperature evolution in the Heat Flux Probe for one elevation and five depths

6 Conclusion

TOPFLOW-PTS Experimental Program is an integral-type program related to the CFD modeling of a PTS two-phase flow configuration. The purpose is to provide a well documented database to perform the validation of CFD models in two-phase flow configurations, and to improve the knowledge and the understanding of the involved key physical phenomena. In this frame, pre-test simulations were carried out with NEPTUNE_CFD code to help in the definition of the test matrix, and possibly of the test procedure, making sure the different key physical phenomena of interest are actually present and can be “caught” by the implemented instrumentation.

These pre-test simulations have pointed out the following main insights:

(i) a gas-liquid flow separation occurs in the ECC injection line when the flowrate becomes lower than a threshold value. It is an actual concern to be accounted for since it strongly modifies the mixing phenomena in the impinging jet area and downstream stratified cold leg area;

(ii) no relevant steady-state can be achieved in an air/water configuration, opposite to the steam/water configuration for which a steady-state stratification can be maintained all along the cold leg thanks to the direct contact condensation of steam;

(iii) for relevant PTS transient tests, the temperature measurements within the implemented *Heat Flux Probe* should allow a assessment of the whole coupled “CFD-Heat conduction” simulations.

7 References

[1] A. Martin *et al.*, "CFD tool for assessment of the Reactor Pressure Vessel Integrity in Pressurized Thermal Shock conditions for life time evaluation. Qualification phases and Thermal Hydraulics study of a safety injection in a PWR plant", Proceedings of NUTHOS 2004, Nara, Japan, October, 2004.

[2] A. Martin *et al.*, "CFD Tools for Assessment of the Reactor Pressure Vessel Integrity in PTS conditions. Thermal Hydraulic methods and main industrial results on the French 900 MWe PWR RPV.", Proceedings of ASME PVP 2009, Prague,Czech Republic, July, 2009.

[3] A. Martin *et al.*, "CFD use in PTS safety analysis. State of art and challenges for industrial applications", Proceedings of NURETH-13 Conference, Kanazawa, Japan, September 27 – October 2, 2009.

[4] P. Péturaud *et al.*, "General overview of TOPFLOW -PTS experimental program", Proceedings of NURETH-14 Conference, Toronto, Ontario, Canada, September 25-30, 2011.

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

[6] C. Chauliac *et al.*, "NURESIM: a European Simulation Platform for Nuclear Reactor Simulation. multi-scale and multi-physics calculations, sensitivity and uncertainty analysis", Proceedings of the Conference on EU Research and Training in Reactor Systems (FISA 2009).

[7] P. Coste, J. Laviéville, J. Pouvreau, C. Baudry, M. Guingo, A. Douce, Validation of the Large interface method of NEPTUNE_CFD 1.0.8 for PTS Applications, CFD4NRS-3 workshop, September 14-16, 2010, Washington DC, USA.

[8] N. Méchitoua *et al.*, "An unstructured Finite Volume solver for two-phase water/vapour flows modeling based on an elliptic-oriented fractional step method", Proceedings of NURETH-10 Conference, Seoul, Korea, October 5-9, 2003.

[9] P. Coste and J. Laviéville, "A Wall Function-Like Approach for-Two- Phase CFD Condensation Modeling of the Pressurized Thermal Shock", Proceedings of NURETH-13 Conference, Kanazawa, Japan, September 27-October 2, 2009.