FLUID MIXING AND FLOW DISTRIBUTION IN THE REACTOR CIRCUIT (FLOMIX-R)

Ulrich Rohde, Thomas Höhne, Sören Kliem, Bengt Hemström[1], John Lillington[2], Martina Scheuerer[3], Timo Toppila[4], Trevor Dury [5], Jan Remis[6], Petr Muhlbauer [7], Ivan Toth [8], Joszef Elter [9], and Yuri Bezrukov[10]

1. Introduction

Coolant mixing inside the nuclear reactor is the most important inherent safety mechanism against boron dilution or overcooling transients and in the case of pressurized thermal shock (PTS) scenarios. In pressurised water reactors (PWR), boron acid is added to the water coolant to compensate the excess reactivity of fresh fuel loadings. Due to different mechanisms or system failures, slugs of low borated water can accumulate in the primary cooling system. This can happen e.g. as a consequence of a small break loss of coolant accident (SB LOCA), when coolant circulation is interrupted and a slug of almost un-borated condensate will accumulate in the cold leg of the primary circuit. During start-up of coolant natural circulation after refilling the primary circuit with the emergency core cooling (ECC) system or by switching on the first main coolant pump (MCP), this slug will be transported into the reactor core causing a significant reactivity insertion by decreasing the concentration of neutron absorber. The mixing of the unborated condensate with borated water in the reactor pressure vessel is in that case the only mitigative mechanism to prevent severe accident consequences. Mixing is relevant not only for nuclear safety, but also for structural integrity. In the case of LOCA, cold ECC water will be injected into the hot primary circuit. When plumes of cold water get in contact with the reactor pressure vessel (RPV) wall, thermal stresses occur, which might endanger RPV integrity. Mixing is even of relevance for normal reactor operation, e.g. to ascertain the coolant temperature distribution at the core inlet in the case of partially switched off MCPs.

In the EC project FLOMIX-R coordinated by Institute of Safety Research, slug mixing and flow distribution in the RPV has been comprehensively investigated experimentally and simulated by using computational fluid dynamics (CFD) tools. Partners from 8 European countries and Russia participated in the project. One objective of the project was to obtain complementary data on slug mixing to understand in sufficient detail, how the slug mixes before it enters the reactor core. Slug mixing experiments have been performed with several 1:5 scaled facilities representing different European reactor types. Additional to slug mixing tests with momentum insertion by starting pumps, experimental results on mixing of fluids driven by density differences were obtained at ROCOM and the FORTUM PTS test facility.

A second objective was to utilise data from steady state mixing experiments and plant commissioning test data to evaluate the primary circuit flow distribution and the effect of thermal mixing phenomena in the context of the improvement of normal operation conditions and structural integrity assessment. Flow distribution data available from commissioning tests (Sizewell-B for PWR, Loviisa and Paks for VVER) were used together with the data from the ROCOM facility as a basis for the flow distribution studies. The test matrix on flow

[1] Vattenfall Utveckling AB, Alvkarleby [2] Serco Assurance, Dorchester, Dorset [3] GRS, Garching [4] Fortum Nuclear Services, Vantaa [5] PSI, Villigen [6] VUJE, Trnava [7] NRI, Rez [8] AEKI, Budapest [9] NPP Paks, Paks [10] EDO Gidropress, Podolsk

distribution and steady state mixing performed at ROCOM comprises experiments with various combinations of running pumps and various mass flow rates in the working loops.

The experimental data were used to contribute to the validation of CFD codes for the analysis of turbulent mixing problems. CFD calculations were accomplished for selected experiments with two different CFD codes (CFX-5, FLUENT). For quality assurance in the CFD code validation, so-called Best Practice Guidelines (BPG) have been applied. The BPG require a minimization of numerical errors and solution errors by systematic grid and time step refinement and sensitivity tests on the impact of uncertainties in the boundary conditions, before the effect of different physical models can be assessed. The applicability of various turbulence modeling techniques was studied for transient and steady state flow.

2. Test facilities

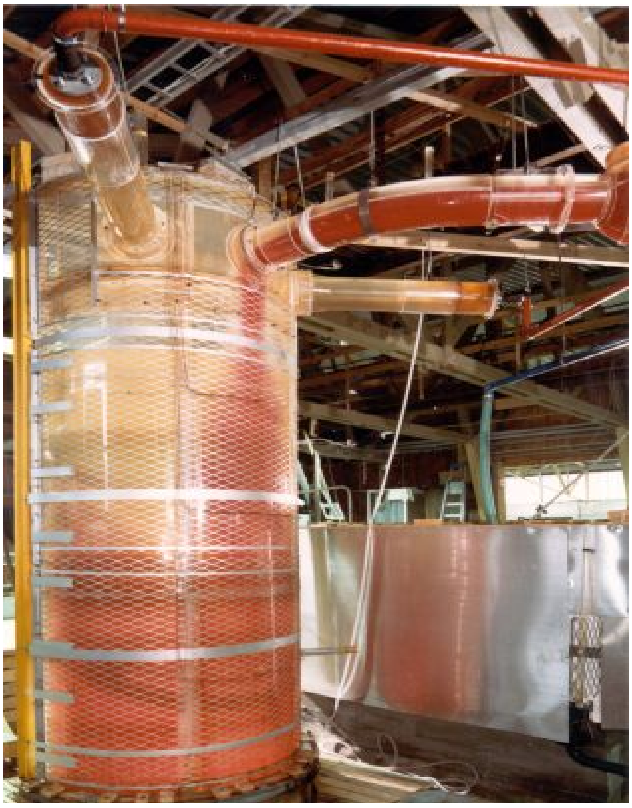
ROCOM (**Ro**ssendorf **Co**olant **M**ixing Model) is a test facility for the investigation of coolant mixing operated with water at room temperature [1]. The facility models a KONVOI type reactor with all details important for the coolant mixing in a linear scale of 1:5. ROCOM is a four-loop test facility with an RPV mock up made of transparent acryl (Fig. 1). Individually controllable pumps in each loop give the possibility to perform tests in a wide range of flow conditions, from natural circulation to nominal flow rate including flow ramps (pump start up). The transparent material of the pressure vessel allows the measurement of velocity profiles in the downcomer by laser Doppler anemometry. Boron concentration and temperature fields are modelled both by the concentration field of a tracer solution (salted water). The normalised tracer concentration is called the mixing scalar.

Fig. 1: View on the test facility ROCOM with the RPV model made from acryl

For the experimental investigation of the coolant mixing, the tracer solution is injected computer-controlled into the cold leg of one of the loops, while the test facility is operated with de-mineralised water. The test facility is equipped with wire-mesh sensors for electrical conductivity measurement [2], which allow the measurement of the transient tracer concentration with high resolution in space and time. Four such sensors are installed in the reactor pressure vessel model with altogether about 1000 single measurement positions and an imaging frequency of up to 200 Hz.

The Vattenfall mixing test facility is a 1:5 scale model of a Westinghouse PWR [3]. The RPV model is partially made of acryl. Components that can be important for mixing have been modelled, for example thermal shields, inlet pipe diffusers, structures in lower plenum, core support plates and core. The investigation of the relative boron concentration is based on salt water tracering and conductivity measurement, too. Conductivity is measured at 181 positions close to the inlet to the core with a sampling frequency of 60 Hz.

The test facility of EDO "Gidropress" [4] is a steel model of the Russian VVER-1000 reactor in a scale of 1:5. One loop with a loop seal, reactor coolant pump simulator and the reactor core with 151 fuel assembly simulators are modelled. Boron concentration change is simulated by a change in temperature (the deborated water slug is simulated by colder water). About 100 thermocouples are placed in the lower part of the downcomer and at the core inlet to study the mixing of flows.



Specific PTS mixing experiments were performed at the Fortum PTS test facility [5]. This facility was a 1:2.56 scaled model of the Loviisa VVER-440 reactor. The facility represented one half of the circumference of the reactor downcomer made of transparent acryl. It included three cold legs, where the middle one was equipped with high pressure injection (HPI) simualtion belonging to the ECCS. Because the choice of the transparent material restricts the tests to a maximum temperature of around 75 °C, an extra buoyancy effect was induced by salt addition to the injected cold HPI water. The relative density difference between HPI and loop flow used in the tests was up to 16 %. The mixing of the HPI water was then observed by measuring temperatures in the downcomer and in the cold leg and visually through the transparent material of the facility.

Fig. 2: View of the Fortum PTS test facility

3. Investigation of momentum controlled and buoyancy driven slug mixing

Fig. 3a shows the time evolution of the mixing scalar at the two sensors in the downcomer in one of the ROCOM slug mixing tests. The mixing scalar distributions are shown over the azimuthally unwrapped downcomer. The position of the loop with the starting up pump is marked by the red arrow. From this visualization it is clearly to be seen, that the de-borated coolant passes around the core barrel instead of flowing directly downstream. Subsequently, at the lower sensor two maximums of the tracer on the “back side” of the downcomer are observed. Therefore, the tracer arrives at the core inlet plane first at positions, which are opposite to the position of the loop with tracer injection.

For the investigation of the influence of density effects, generic experiments have been carried out at the ROCOM test facility. The objective of these experiments was to find the conditions for transition from momentum controlled mixing, as it is typical for pump start-up scenarios, to buoyancy driven mixing, being relevant for PTS scenarios and natural circulation re-start after LOCA. It is expected, that density differences can be neglected, if the flow rates are sufficiently high. Because the ROCOM facility cannot be heated up, the necessary density differences were simulated by adding sugar (glucose) to the water that is injected into the cold leg. To observe the mixing of the ECC water fed into the cold leg by the HPI system, this water was tracered by small amounts of sodium chloride, as in previous experiments. The maximum density difference created by the addition of glucose was 10%.

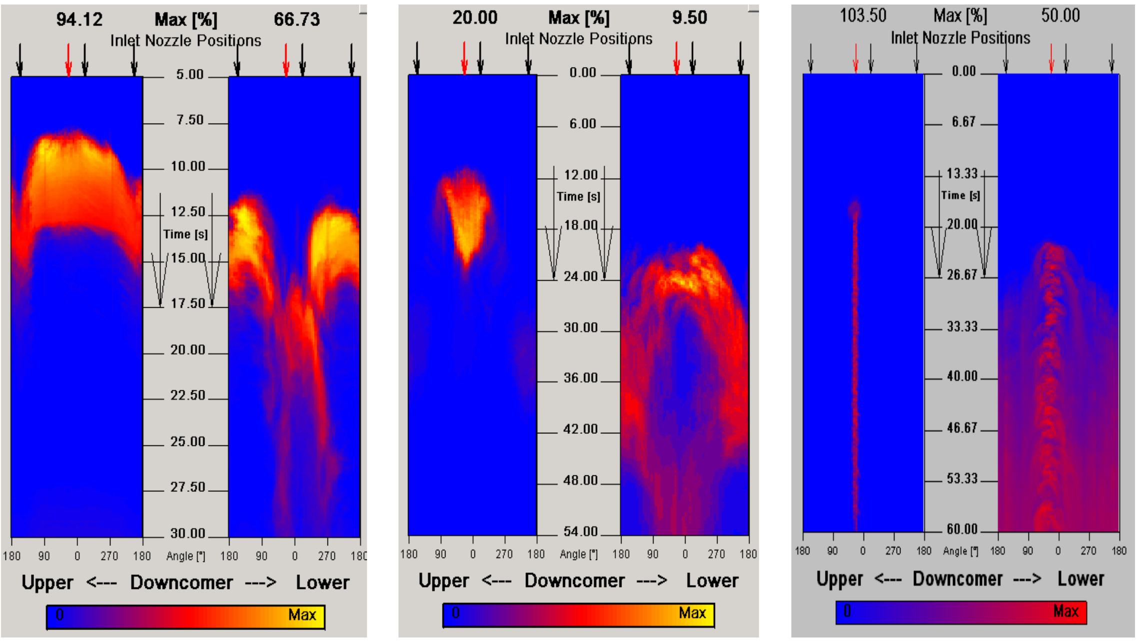


Fig. 3: Time evolution of the mixing scalar unwrapped over the downcomer

a) nominal flow rate, no density diff.　　*b) 10 % flow rate 10 % density diff.*　　*c) no flow rate, 10 % density diff.*

An unwrapped view of the time evolution of the tracer concentration measured at the two downcomer sensors in the experiment with 10% density difference and no flow in the injection loop (Fig. 3c) shows, that the sector covered by the ECC water is very small in the upper dowcomer in this case. The ECC water falls down straightly and passes the sensor in the lower part of the downcomer just below the inlet nozzle of the working loop. This mixing pattern is completely different from that one observed in the case of pump-start-up shown in Fig. 3a. Fig. 3b shows the mixing scalar behaviour in an experiment with high density difference, but low flow rate. The spreading of the mixing scalar into two streams is observed only in the lower downcomer. Based on these observations, the experiments with density differences can be divided into three groups: density dominated flow (see Fig. 3c), momentum dominated flow (see Fig. 3a) and the transition region (see Fig. 3b). The conditions at the inlet into the downcomer were used to calculate Froude-numbers of the experiments according to the following formula:

$$Fr_{DC} = \frac{v_{in}}{\sqrt{g \cdot H \cdot \frac{\rho_{in} - \rho_a}{\rho_{in}}}} \tag{1}$$

where v_{in} is the velocity at the reactor inlet (combined loop and ECC flow), g is the gravitational acceleration, H is the height of the downcomer, ρ_{in} the density of the incoming flow, calculated with the assumption of homogeneous mixing between ECC and loop flow, and ρ_a the density of the ambient water in the downcomer. All experiments, identified as density dominated are characterised by Froude numbers less than 1.0, and all momentum dominated cases are found in the region of $Fr > 1.0$. Therefore, $Fr = 1$ is a critical Froude numbers separating the two flow regimes of momentum dominated and buoyancy driven flow.

The observations on density driven mixing have been confirmed qualitatively in the experiments performed at the Fortum PTS facility. Applying the same Froude scaling as it is given in equ. (1), all Fortum PTS tests are located in the density driven mixing range.

4. CFD code validation

A tremendous work on CFD code validation was performed within the FLOMIX-R project. The commercial CFD codes CFX-4, CFX-5 and FLUENT-6 have been used. Systematic code validation based on so-called Best Practice Guidelines (BPG) was focussed mainly on a number of benchmark cases from the steady-state mixing experiments, slug mixing tests and experiments with density differences. The ERCOFTAC BPG [6], which have been specified for nuclear reactor safety calculations within the ECORA project [7] were used for quality assurance of the validation calculations. The BPG are built on the concept of an error hierarchy. The different types of errors in CFD simulations are divided into the two main categories:

- Numerical errors, caused by the discretisation of the flow geometry and the model equations, and by their numerical solution
- Model errors, which arise from the approximation of physical processes by empirical mathematical models

This concept implies that numerical errors are quantified and reduced to an acceptable level, before comparison with experimental data is made. The BPG contain a set of systematic procedures for quantifying and reducing numerical errors. The knowledge of these numerical errors is a prerequisite for the proper judgement of model errors. Numerical errors are minimised by optimising the computational mesh, numerical schemes, convergence criteria and time step. Another kind of errors are uncertainties arising from insufficient information about the problem definition and set-up, like boundary positions, boundary conditions and internal geometry modelling. These uncertainties can be quantified by sensitivity analyses. Turbulence models are most relevant for physical errors.

Fig. 4 compares the CFD solution and experiment for the time-averaged mixing scalar distribution at the core inlet in a ROCOM steady-state mixing test with running pumps. The best agreement with the experiment was achieved, when even for the steady state a transient calculation was performed to reproduce turbulent fluctuations of the velocity and tracer concentration field observed in the experiment. Note, that the time-averaged distributions are presented on Fig. 4. A tetrahedral mesh with about 7 million elements comprising the detailed resolution of the internal structures, including the sieve drum in the lower plenum with 410 orifices was used. Standard K,ε and Shear Stress Transport (SST) K,ω turbulence models were applied. The different turbulence models provide very similar results.

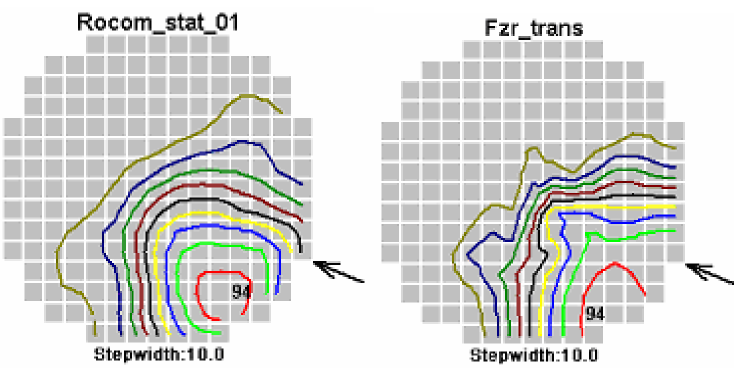


Fig. 4: Comparison between CFD solution (right) and experimental data (left)

The following conclusions were drawn from the CFD validation work:

- For correct description of inlet boundary conditions, at least a part of the cold leg should be modelled.
- Internal geometry should be modelled as detailed as possible due to limitations of the porous body approach. This requires a continuous progress in pre-processors.
- Concerning turbulence models, first order models like K-ε or SST K-ω can be recommended. For buoyancy driven mixing, better results have been obtained with Reynolds stress models. However, no final conclusions can be drawn, because BPG solutions could not be achieved in all cases.

5. Summary

A new quality of research in flow distribution and turbulent mixing inside the RPV of nuclear reactors has been achieved in the FLOMIX-R project. Experimental data on slug mixing with enhanced resolution in space and time have been gained from various test facilities covering different geometrical and flow conditions. The basic understanding of momentum controlled mixing in highly turbulent flow and buoyancy driven mixing in the case of relevant density differences between the mixing fluids has improved significantly. A higher level of quality assurance in CFD code validation has been achieved by consequently applying BPG.

References

[1] Prasser, H.-M., G. Grunwald, T. Höhne, S. Kliem, U. Rohde, F.-P. Weiss: “Coolant mixing in a pressurised water reactor: deboration transients, steam-line breaks, and emergency core cooling injection”, Nuclear Technology Vol. 143 (July 2003), pp. 37-56

[2] Prasser, H.-M., Böttger, A., Zschau, J.: “A New Electrode-Mesh Tomograph for Gas Liquid Flows,” Flow Measurement and Instrumentation, 9, 111-119 (1998).

[3] Alavyoon, F., Hemström, B., Andersson, N. G., Karlsson; R. I.: “Experimental and Computational Approach to Investigating Rapid Boron Dilution Transients in PWRs,” CSNI Specialist Meeting on Boron Dilution Reactivity Transients, State College, PA, USA, October 18-20, (1995).

[4] Logvinov S.A., Ulyanovsky V.N., Bezrukov Yu.A., Kozlov A.N.: “Mixing of coolant with different boron concentration at the VVER-1000 core inlet during RCP start-up”, Proc. ANNUAL MEETING ON NUCLEAR TECHNOLOGY 2000, Bonn, 22-24 May 2000.

[5] Tuomisto, H.: “Thermal-hydraulics of the Loviisa reactor pressure vessel vercooling transients”, Imatran Voima Oy, Research report IVO-A-01/87, 1987.

[6] ERCOFTAC Best Practice Guidelines, see http://imhefwww.epfl.ch/lmf/ERCOFTAC/

[7] Menter, F. et al.: CFD Best Practice Guidelines for CFD Code Validation for Reactor Safety Applications, Deliverable D01 of the ECORA project, February 2002

Acknowledgement

The project this paper is based on was funded by the European Commission under contract number FIKS-CT-2001-00197.