Coolant mixing in a PWR - deboration transients, steam line breaks and emergency core cooling injection - experiments and analyses

Forschungszentrum Rossendorf, Institute of Safety Research
P.O.B. 510119 D-01314 Dresden, Germany
Tel: (+49) 351 260 3460, Fax: (+49) 351 260 3460, Email: prasser@fz-rossendorf.de

Abstract - The reactor transient caused by a perturbation of boron concentration or coolant temperature at the inlet of a Pressurized Water Reactor (PWR) depends on the mixing inside the reactor pressure vessel. Initial steep gradients are partially lessened by turbulent mixing with coolant from the unaffected loops and with the water inventory of the reactor pressure vessel. Nevertheless the assumption of an ideal mixing in the downcomer and the lower plenum of the reactor leads to unrealistically small reactivity inserts. The uncertainties between ideal mixing and total absence of mixing are too large to be acceptable for safety analyses. In reality, a partial mixing takes place. For realistic predictions it is necessary to study the mixing within the three-dimensional flow field in the complicated geometry of a PWR. For this purpose a 1:5 scaled model (the ROCOM facility) of the German PWR KONVOI was built. Compared to other experiments, the emphasis was put on extensive measuring instrumentation and a maximum of flexibility of the facility to cover as much as possible different test scenarios. The use of special electrode-mesh sensors together with a salt tracer technique provided distributions of the disturbance within downcomer and core entrance with a high resolution in space and time. Especially the instrumentation of the downcomer gained valuable information about the mixing phenomena in detail. The obtained data was used to support code development and validation. Scenarios investigated are: (1) Steady-state flow in multiple coolant loops with a temperature or boron concentration perturbation in one of the running loops. (2) Transient flow situations with flow rates changing with time in one or more loops, such as pump start-up scenarios with deborated plugs in one of the loops or onset of natural circulation after boiling-condenser-mode operation. (3) Gravity driven flow caused by large density gradients, e.g. mixing of cold emergency core cooling water entering the RPV through the ECC injection into the cold leg. The experimental results show an incomplete mixing with typical concentration and temperature distributions at the core inlet which strongly depend on the boundary conditions. CFD calculations were found to be in good agreement with the experiments.

I. INTRODUCTION

A perturbation of boron concentration or coolant temperature at the inlet of a Pressurized Water Reactor (PWR) may lead to a reactivity excursion. The induced power peak and the released fission energy depend on the inserted reactivity. As a matter of rule the perturbation has to be expected not in all loops of the primary circuit in the same time – the probability of an appearance in one of the loops only is much higher. On the way from the reactor inlet to the core entrance this water comes into contact with coolant arriving from the unaffected loops and with the water inventory of the reactor pressure vessel. The boron concentration and temperature distributions at the core inlet are therefore the result of a turbulent mixing process in the flow path of the reactor downcomer and the lower plenum.

3D neutron kinetic computer simulations have shown quite early, that a uniform distribution of the perturbation at the core inlet, i.e. the assumption of an ideal mixing, does not deliver conservative results. In fact, the predicted power peak was about five times lower compared to analyses where a total absence of mixing was assumed [1]. In this work, attempts were initially made to implement stationary mixing matrixes in a 3D neutron kinetics code (DYN3D). The remaining uncertainties were too large to be acceptable for safety analyses. There are numerous postulated accident scenarios, where in case of uniform mixing the core survives without consequences or even without reaching criticality, while neglect of the mixing leads to the prediction of severe fuel rod failures.

In reality, a partial mixing takes place. In the result, the real distribution of fluid condition at the core entrance lies in-between the two extreme cases both concerning shape and amplitude of the perturbation. To make realistic predictions about the consequences it is necessary to study the coolant mixing, which is a complicated three-dimensional fluid dynamic problem. Modern CFD codes running on powerful computers became available to model the coolant flow in the complex geometry of a PWR a couple of years ago.
Nevertheless the use of CFD for this complicated task is a challenge still today. Due to the high safety relevance of the coolant mixing phenomenon it is necessary to validate computer codes and to verify computational results using experimental data. This explains the vivid scientific interest in coolant mixing experiments. In our own experiments presented in this paper, we put emphasis on flexibility of the test facility to cover as much as possible different test scenarios, and, above all, on measuring instrumentation in order to meet the requirements of code development and validation as much as possible by providing parameter distributions with a high resolution in space and time.

Since the characteristic Reynolds numbers for the large dimensions of the reactor pressure vessel (RPV) are very high, turbulent mixing dominates over molecular diffusion. For this reason in a given velocity field a temperature perturbation and a boron concentration perturbation behave in the same way, however in case of high temperature differences density gradients may cause a feedback to the velocity field by gravity effects. From the point of view of fluid dynamic boundary conditions, the mixing scenarios relevant for the safety of PWRs can be subdivided into three large groups:

1. Steady flow in multiple coolant loops with a temperature or boron concentration perturbation in a part of the running loops. The flow may be driven either by running main coolant pumps or by steady natural circulation. A typical case is the mixing of over-cooled water arriving from one of the steam generators in case of a main steam-line break.

2. Transient flow situations with flow rates changing with time in one or more loops. Also here the flow may be driven either by coolant pumps or by natural circulation. Prominent examples are: (a) inadvertent start-up of the main coolant pump in a situation where boron-free coolant was unintentionally accumulated in the corresponding loop, e.g. by a malfunction of the auxiliary feed water system during a shut-down of the reactor or due to heat exchanger tube leakage in a steam generator, (b) onset of natural circulation in a late phase of a small-break loss-of-coolant accident after an accumulation of boron free condensate by boiling-condenser mode operation.

3. Gravity driven flow caused by large density gradients. This phenomenon is observed only if the main coolant pumps are not working, otherwise the momentum driven flow induced by the pumps is always dominating and density effects can be neglected. Gravity effects are characteristic, for example, for the mixing of cold emergency core cooling water entering the RPV through the ECC injection into the cold leg. Due to the higher density of the ECC water, the formation of a streak of cold water flowing down in the downcomer is observed. This causes thermal loads on the hot reactor pressure vessel. Beside the prediction of the parameters at the core inlet, the exact knowledge of the spatial distribution of the ECC water is essential for the assessment of the pressure vessel behavior (Pre-stressed Thermal Shock = PTS).

The literature reports about a number of test facilities to study coolant mixing, each considering the geometrical specifics of various reactor types [2-9]. In case of [2-5], the facilities were solely designed to study the inadvertent pump start-up, i.e. they dispose of a circulation pump only in one of the loops; the other loops were just passive ones. Other test facilities were dedicated to mixing of cold emergency core cooling (ECC) water injected into the cold leg of a PWR [6-9]. In order to study coolant mixing inside the reactor pressure vessel of a German type PWR the test facility ROCOM (Rossendorf Coolant Mixing Model) was built and operated during the last three years. The facility represents a KONVOI type PWR (1300 MW<sub>el</sub>) developed by Siemens KWU in a linear scale of 1:5.

II. ROCOM TEST FACILITY

ROCOM consists of a reactor pressure vessel model (Fig. 1) with four inlet and four outlet nozzles. Since it was planned to use the facility for a wide range of assumed mixing scenarios, ROCOM was equipped with four fully functioning loops (Fig. 2), i.e. it has four
circulation pumps, which are driven by motors with computer controlled frequency transformers. In this way, a wide variety of flow rate regimes, such as four-loop operation, operation with pumps off, simulated natural circulation modes and flow rate ramps can be realized. For natural circulation conditions the corresponding pumps are operated at low rotation speed by means of the frequency transformer system. Beginning from the bends in the cold legs which are closest to the reactor inlet, the geometrical similarity between model and original reactor is respected until the core inlet. The core itself is excluded from the similarity, i.e. cross mixing within the core is not investigated. The reactor model is manufactured of acrylic glass (Perspex, see Fig. 3). A total view of the test facility is given in Fig. 4. The reactor model exactly follows the geometry of the original PWR with respect to the design of the nozzles (diameter, surface curvature, diffuser parts), the characteristic extension of the downcomer cross section below the nozzle zone, the so-called perforated drum in the lower plenum and the design of the core support plate with the orifices for the coolant.

Fig. 2 ROCOM test facility with four loops and individually frequency controlled circulation pumps

The flow rate in the loops is scaled according to the transit time of the coolant through the model. Since the geometrical scale is 1:5, the transition time of the coolant is identical to that of the original reactor, when the coolant velocity is scaled down by 1:5, too. The nominal flow rate in ROCOM is therefore 185 m³/h per loop.

As a result of the scaling and the increased viscosity of cold water compared to that of the hot coolant, the Reynolds numbers (Re) in the model are lower than those in the original reactor by factor of 190.

At the maximum flow rate (300 m³/h) delivered by the pumps, a Re ratio of 100 can be achieved. However, the Re numbers (of the order of 10^5 in the ROCOM facility) correspond to highly turbulent flow conditions, thus both velocity fields and mixing condition are supposed to be independent of the mass flow rate (if density gradients are absent). This assumption is supported by own measuring results obtained for flow rates varied in the feasible range, by scale-up investigations using a CFD code [10], as well as by findings of Dräger [11, 12], who took advantage from real-scale experiments carried out at Russian type VVER-440 reactors.
To vary the flow rate the frequency of the power supply for the pumps is controlled. Computer controlled frequency transformers allow to run the pumps according to predefined time dependencies of the flow rate, given as data maps for each loop individually.

Fig. 3 Photo of the reactor pressure vessel model of ROCOM made of acrylic glass (Perspex)

The facility is operated with water at room temperature and ambient pressure. Since the coolant mixing is mainly based on turbulent dispersion it is possible to use a tracer substance to model differences of either boron concentration or coolant temperature. In case of ROCOM the coolant in the disturbed loop was labeled by injecting a sodium chloride solution into the main coolant flow upstream of the affected reactor inlet nozzle. For this purpose, loop 1 of ROCOM was equipped with a mixing device, which distributes the tracer solution uniformly across the cross section of the main circulation pipe. Start and stop of the injection process was controlled by magnetic valves. The concentration distribution of the tracer within the reactor model is measured by electrical conductivity sensors.

In case of the experiments on ECC injection, the mixing device was replaced by an injection nozzle, which was geometrically similar to the original KONVOI reactor. There, a horizontal injection line is used, which has a diameter of 0.2 m (at ROCOM $\varnothing$ 40 mm). It is connected to the cold leg under an angle of 45 deg. The higher density of the cold ECC water was simulated by adding sugar (glucose), since density gradients cannot be created by temperature differences, because the facility cannot be heated up. Fortunately, the viscosity of glucose solution becomes large only at concentrations, where the relative density increase is well above the 10 % necessary for the experiments. A sugar solution with the corresponding density of 1100 kg/m$^3$ has a viscosity which is still just by factor of about 3 higher than that of pure water. The tracer can therefore still be envisaged as a low-viscous fluid.

Fig. 4 General view of the ROCOM test facility

III. INSTRUMENTATION

The tracer distribution in the reactor model was observed by so-called electrode-mesh sensors, which measure the distribution of the electrical conductivity over the cross section of the flow duct. Sensors of this kind were introduced by Johnson [13], who used them to measure the integral gas fraction in the cross section. Reinecke et al. [14] presented a tomographic device visualizing sequences of gas fraction distributions with a rate of about 100 frames per second. Our own development was aimed at a direct conductivity measurement between pairs of crossing wires to avoid tomographic reconstruction algorithms [15] and to reach a time resolution of up to 10 000 frames per second [16]. Two crossing grids of electrodes insulated from each other are placed across the flow duct. The electrodes of the first grid (transmitter electrodes) are supplied with short voltage pulses in a successive order. The currents arriving at the electrodes of the second grid are recorded (receiver electrodes). After a complete cycle of transmitter activation a complete matrix of local conductivities is obtained. Special methods of signal acquisition [15] guarantee that each value of the matrix depends only on the local conductivity in the immediate vicinity of the corresponding crossing point between transmitter and receiver electrodes.

Measured local conductivities are afterwards related to reference values. The result is a so-called mixing scalar that characterizes the instantaneous share of coolant originating from the disturbed loop (i.e. where the tracer is injected) at a given location inside the flow field. The scalar is dimensionless. Assuming
similarity between tracer field and the temperature and boron concentration fields it can be used to apply the experimental results to the original reactor. The reference values correspond to the unaffected coolant (index 0) and the coolant at the disturbed reactor inlet nozzle (index 1). The difference between the two reference values is the magnitude of the perturbation. The mixing scalar $\Theta$ is defined as follows:

$$\Theta_{x,y,z,t} = \frac{\sigma_{x,y,z,t} - \sigma_0}{\sigma_1 - \sigma_0} = \frac{T_{x,y,z,t} - T_0}{T_1 - T_0} = \frac{C_{B,x,y,z,t} - C_{B,0}}{C_{B,1} - C_{B,0}}$$

Here, $\sigma$ is the electrical conductivity, $T$ is the temperature and $C_B$ the boron concentration. Which of the two parameters temperature or boron concentration is represented by the measured mixing scalar depends only on the right choice of the reference values and the set-up of the boundary conditions in the experiment. If the tracer is for example injected corresponding to temperature changes in a given scenario, the mixing scalar represents the temperature. Additional measures have to be taken, if the velocity field would be influenced by a feed-back from the temperature-dependent fluid density. This is the case in the tests modeling the cold ECC water injection described later on.

**Fig. 5** Mesh sensor for measuring tracer distributions upstream of the reactor inlet nozzle (16x16 electrodes)

Mesh sensors are placed at four positions of the flow path. The first sensor (Fig. 5) is flanged to the reactor inlet nozzle (Fig. 1) in loop 1. It is aimed at the observation of the distribution at the reactor inlet, which is used as reference distribution for calculating mixing scalars. It has 2 x 16 electrode wires, which are crossing under an angle of 90 deg, i.e. the measuring matrix has the dimension of 16 x 16. The pitch of the wires is 8.9 mm, the total amount of measuring points is 216, since some of the crossing points are outside the circular cross section of the pipe.

The second and the third sensors are located in the downcomer (Fig. 1 and Fig. 6). They represent 4 x 64 measuring positions, i.e. the azimuthal distribution of the tracer concentration is measured at 64 angular positions with a pitch of 5.625 deg. Over the radius there are 4 measuring positions with a radial pitch of 13 mm. The downcomer sensors consist of radial fixing rods with orifices for four circular electrode wires. Rods and wires are separated electrically by small ceramic insulation beads. The rods are acting as radial electrodes, i.e. each rod corresponds to an azimuthal measuring position.

**Fig. 6** Mesh sensor for the downcomer (4x64 measuring points)

The fourth sensor is integrated into the core support plate (Fig. 7). 2 x 15 electrode wires are arranged in a way that the wires of the two planes cross in the centers of the coolant inlet orifices of each fuel element. In this way, the tracer concentration is measured at each fuel element inlet.
All four sensors are connected to a signal acquisition unit disposing of measuring matrix comprising of a 32 x 32 points. The sensors occupy different quadrants of this matrix. In total, about 1000 measuring points are recorded, the measuring frequency is 200 Hz. In most of the cases 10 successive measurements were averaged and the result was stored with a frequency of 20 Hz, because the characteristic frequency of the observed phenomena were not requiring a higher sampling frequency.

IV. MIXING UNDER STEADY FLOW CONDITIONS

The tests for steady flow conditions were performed with different numbers of operating pumps including the normal operation regime with all four pumps running [17]. The pumps were working at constant rotation speed. The perturbation was modeled by injecting the tracer into loop 1, the mixing device guaranteed an equal distribution over the cross section of the main circulation pipe. In the tests presented below, the tracer was injected over a long period (=10 s), which was sufficient to establish a quasi-stationary concentration field. The injection was stopped just before the tracer cloud reappeared at the reactor inlet after completing an entire circulation in the primary circuit. The average conductivity measured by the sensor at the reactor inlet during the quasi-stationary period was used as reference value $\sigma_1$. The second reference value $\sigma_0$ is the conductivity measured before the injection started.

![Fig. 8 Time averaged tracer concentration distribution at all four sensors during quasi steady-state flow, scaled nominal flow rate (185 m³/h in all four loops), perturbation in loop 1 (red arrow)](image)

The distributions measured by the sensors in the downcomer and at the core entrance were also averaged over the quasi-stationary period. In the result, average mixing scalars, i.e. so-called mixing coefficients were obtained, which show the average share of coolant arriving from the disturbed loop at the given measuring location (Fig. 8). In the presented case all main circulation pumps are delivering identical flow rates. This corresponds to an early stage of a main steam-line break scenario and the tracer represents over-cooled coolant coming from the disturbed steam generator. In this case the labeled water occupies a sector of 90 deg at the upper downcomer sensor. The azimuthal position of this sector is shifted from the angular position of the inlet nozzle (22.5 deg) towards 45 deg corresponding to the symmetry conditions. At the lower downcomer sensor, the slopes of distribution become more smooth caused by the action of turbulent mixing at the boundary of the labeled water stream. At the core entrance, a characteristic distribution is found, which shows a maximum near the symmetry position of 45 deg at the side of the azimuthal position of the affected inlet nozzle. The maximum value of the mixing coefficient reaches 91 %, i.e. there is a fuel element receiving almost the full amplitude of the perturbation. There is almost no tracer found in large regions opposite to the affected loop. Nevertheless, the effect of the perturbation on the reactivity is much smaller than in case of assumed absence of mixing due to the smooth slopes of the distribution.
On the first glance, these results seem to prove that the flow field is similar to what was found in the very comprehensive mixing experiments carried out by Dräger [11, 12] for the Russian type pressurized water reactor VVER-440. There, it was found that the flow field in the downcomer can be approximated by a potential flow with quite straight streamlines in the downcomer. A closer look to the transient behavior during the arrival of the tracer slug shows that the flow conditions in the downcomer are more complicated (Fig. 9). Tracer arrives at first at two separate azimuthal positions. Only later the two streams fuse together. This is caused by secondary vortices in the downcomer, which are induced by the widening of the downcomer in the region below the inlet nozzles. There, the width of the downcomer is increased by the reduction of the wall thickness of the reactor pressure vessel (see Fig. 1). The resulting diffuser angle of 18 deg is too big to avoid flow separation.

Secondary vortex flow is the result, causing a non-uniform velocity field at the lower end of the downcomer, which was found also by LDA measurements (Fig. 10). The vortices are rotating in the plane of the unwrapped downcomer, they are intermittently changing their strength, which results in strong fluctuations of the tracer concentration at the periphery of the affected sector of the core entrance (Fig. 11). The assumption of a potential flow can therefore not be applied for this type of reactor.

Secondary vortex flow is the result, causing a non-uniform velocity field at the lower end of the downcomer, which was found also by LDA measurements (Fig. 10). The vortices are rotating in the plane of the unwrapped downcomer, they are intermittently changing their strength, which results in strong fluctuations of the tracer concentration at the periphery of the affected sector of the core entrance (Fig. 11). The assumption of a potential flow can therefore not be applied for this type of reactor.

Secondary vortex flow is the result, causing a non-uniform velocity field at the lower end of the downcomer, which was found also by LDA measurements (Fig. 10). The vortices are rotating in the plane of the unwrapped downcomer, they are intermittently changing their strength, which results in strong fluctuations of the tracer concentration at the periphery of the affected sector of the core entrance (Fig. 11). The assumption of a potential flow can therefore not be applied for this type of reactor.
on the back side of the downcomer are observed. In the result the tracer appears at the core entrance at the side nearly opposite to the affected inlet nozzle. A sequence of instantaneous mixing scalars at the core entrance is shown in Fig. 18a together with theoretical results. The maximum value reached 59 % in this test. It strongly depends on the volume of the initial slug of deborated coolant, i.e. the amplitude of the deboration of the core decreases, if the slug is smaller. Similar results were obtained in tests where the flow rate was close to natural circulation conditions.

Fig. 12 Flow rate in loop 1 and counter flow in loops 2-4 during pump start-up, averaged tracer concentration at the reactor inlet nozzle

![Graph showing flow rate and tracer concentration](image)

VI. SIMPLIFIED MATHEMATICAL MODEL

A simplified mathematical model (SAPR - Semi-Analytical Perturbation Reconstruction) was developed to generalize experimental results using some linear properties of the transport equation for the temperature respectively the boron concentration [19]. By means of a superposition of data measured at the core entrance for short tracer pulses (quasi Dirac impulses) generated at the reactor inlet, the length and the amplitude of the disturbance can be varied off-line, i.e. without repeating the experiments. Since the flow rate in the loop is changing, the system response is depending from the moment of injection. Therefore, the Dirac impulses have to be put in different points on the flow rate ramp.

The method was checked using the experimental data of the test described in the previous chapter. The applied input pulses and the experimentally obtained responses are shown in Fig. 14. Additional intermediate Dirac-responses were derived by interpolation. The time history of the tracer concentration during the long injection (Fig. 12) was reconstructed by superposing these Dirac perturbations (Fig. 15). The superposition of the corresponding time responses is in good agreement to the directly measured one (Fig. 18b). Among others, the method can be used to correct the decrease of the tracer concentration in the experimental slug. Due to the fact that tracer solution was injected with a constant
flow rate into the main flow which was increasing in
time, the concentration in the experimental slug was
decreasing (corresponding to increasing boron
concentration) during the injection period (Fig. 12). In
practice the slug is more or less homogeneous. Another
application of the method is the coupling of 3D neutron
kinetic codes with thermal hydraulic system codes.

VII. APPLICATION OF THE SAPR MODEL TO A
HYPOTHETICAL BORON DILUTION EVENT

The SAPR model has been coupled with the 3D
reactor dynamics code DYN3D [20] to provide realistic
boron concentration or temperature distributions as
boundary conditions for transient analyses. The
DYN3D code has been developed in the Institute of
Safety Research of the Forschungszentrum Rossendorf.
DYN3D has an extensive verification and validation
basis [21, 22] and is widely used for the analysis of
reactivity initiated accidents in light water reactors with
Cartesian and hexagonal fuel assembly cross section
geometry [23, 24].

The analysis presented here has been carried out
for the beginning of an equilibrium fuel cycle of a
generic four-loop pressurized water reactor. The
macroscopic cross section library needed for the core
calculations has been generated by the 2D neutron
transport code HELIOS. The library contains cross
section sets dependent from burn-up and the thermo-
hydraulic feedback parameters in a range of variation
being relevant for the transient under consideration. The
reactor is assumed at hot zero power in a subcritical
state. The Xenon- and Samarium-distribution
 corresponds to the full power state. All control rods are
inserted, except the most effective, which sticks at fully
withdrawn position. The considered boron dilution
scenario is based on the analysis of the restart of the
first main coolant (MCP) pump in a PWR after a steam
generator tube rupture accident [25]. It was assumed in
this analysis, that a slug of boron-free coolant has been
created in the main circulation loop and is driven into
the core by start-up of the first MCP. The coolant in the
lower plenum has a temperature of 192 °C and a boron
content of 2200 ppm. The temperature of the deborated
slug is 210 °C. The initial subcriticality of this state
(before the restart of the circulation) is determined
with -7787 pcm.

The MCP reaches its full mass flow rate about 15 s
after the start-up. The boron front reaches the core entry
about 12 s after switching-on the MCP. The boron
dilution in the reactor core due to the slug causes a
super prompt critical reactivity insertion leading to a
very short power pulse with a magnitude of more than
7000 MW (Fig. 16).

It is limited due to the strong, practically promptly
acting Doppler feedback of the fuel temperature. The
half width of the power peak was so small that a
significant enhancement of the coolant temperature did
not occur at that moment. Due to the further deboration, the positive reactivity insertion is continued after the first power peak. Typical secondary power peaks are observed. As can be seen from figure 17, the radial power distribution over the reactor core is very non-uniform. At the location of the power maximum (near to the position of the sticking out control rod), coolant boiling with a maximum void fraction of up to 70% occurs for a short time. However, no occurrence of heat transfer crisis was obtained, so that the cladding temperatures remained below 260 °C. Safety margins are not violated.

VIII. CFD CALCULATIONS

The code CFX-4 [26] is used for complementary calculations in test facility and original reactor geometry. The coolant flow is assumed to be incompressible. The standard k, ε turbulence model is used. Inlet boundary conditions (velocity, temperature, boron concentration etc.) are set at the inlet nozzles. The boundary conditions at the core outlet are pressure controlled. For the description of temperature, boron respectively salt tracer concentrations, passive scalar fields are used. Special attention was paid to the exact representation of the nozzle region with the curvature radii at the four inlet nozzles, the passes of the hot legs through the downcomer and the diffuser-type extension of the downcomer below the nozzles. Due to the great importance of this region for the coolant mixing, the grid is very dense in these regions. The generated grid consists of about 450000 nodes. The core support plate, the core and the perforated drum are modeled as a porous region. The porosity is determined by relating the area of orifices to the total area of the sieve.

The body forces taking into account distributed friction losses in the sieve plate are determined to obtain equal pressure drop over the structures as they are derived from the detailed construction. Steady state calculations with at least 1000 iterations last a few hours, transient calculations a few weeks on an SGI Origin 200 workstation.

**Fig. 18 Time sequences of selected instantaneous mixing scalar distributions at the core entrance during pump start-up, 36 m³ deborated coolant in loop 1, maximum indicated with bold number at corresponding fuel element position**
Fig. 19 Time history of the instantaneously observed maximum mixing scalar at the core entrance during the pump start-up test

Fig. 20 Stream lines in the reactor pressure vessel calculated by CFX-4 for the case of pump start-up in one of the loops

The result of a post-test calculation of the pump start-up experiment described above is given in figure 18c. There are quantitative deviations of few percent concerning the time history of the instantaneous maximum mixing scalar (Fig. 19). Larger deviations are found at a number of fuel element positions mainly in the central region of the core. Furthermore, the calculated distributions show a certain delay ($\approx 1\text{ s}$) compared to the experiment. The calculation results are in good correspondence to the experiment concerning the main mixing phenomena, such as azimuthal position of tracer appearance and the shape of the found distributions. In general it was found that CFX is reproducing most of the measured results with a satisfactory reliability in case of the single-phase flow characteristic for the investigated mixing scenarios, as long as density effects are absent. The latter case requires additional code development and validation activities.

CFD calculations are powerful tools to explain fluid dynamic effects by plotting stream lines. This was done in figure 20 for the pump start-up scenario. It is clearly visible that the coolant has the tendency to flow around the core barrel. The flow is divided into two parts, which meet again at the side opposite to the affected inlet nozzle. This explains the shape of the tracer distributions presented in figure 18.

IX. ECC WATER INJECTION WITH DENSITY EFFECTS

During emergency core cooling the higher density of the ECC water leads to a streak of cold water that is flowing downwards in the downcomer. This causes thermal loads on the RPV. Furthermore, in the case of inadvertent injection of low borated ECC water, a boron dilution transient would be initiated. The transient is then determined by the resulting boron concentration distribution at the core inlet.

Due to the fact, that the test 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, this water was again labeled by small amounts of sodium chloride, enhancing the conductivity. Generating density differences by high salt concentrations is not possible, because the measurement system is very sensitive and would be saturated at high salt concentrations.

The goal of the experiments presented in the current work was the generic investigation of the influence of density differences between the primary loop inventory and the ECC water on the mixing in the downcomer. To separate the density effects from the influence of other parameters, a constant flow in the loop with the ECC injection nozzle was assumed in this study. The mass flow rate was varied between 0 and 15% of the nominal flow rate, i.e. it was kept in the magnitude of natural circulation. The other pumps were switched off. The density difference between ECC and
loop water was varied between 0 and 10%. Figure 21 summarizes the boundary conditions of the experiments. Altogether 20 experiments have been carried out (dots in figure 21). In all experiments, the volume flow rate of the ECC injection system was kept constant at 1.0 l/s. The normalized density is defined as the ratio between ECC water density and density of fluid in the circuit. All other boundary conditions are identical.

The experiments without density effects serve as reference experiments for the comparison. Fig. 22 shows the time evolution of the tracer concentration measured at the two downcomer sensors in an unwrapped view. The downwards directed red arrow indicates the position of the loop with the running pump, in that case delivering 10% of the nominal flow rate. At the upper downcomer sensor, the ECC water (injected in each experiment from $t = 5$ to $t = 15$ s) appears directly below the inlet nozzle. Due to the momentum created by the pump, the flow entering the downcomer is divided into two streams flowing right and left in a downwards directed helix around the core barrel. At the opposite side of the downcomer, the two streaks of the flow fuse together and move down through the measuring plane of the lower downcomer sensor into the lower plenum. Almost the whole quantity of ECC water passes the measuring plane of the lower downcomer sensor at the side nearly opposite to the azimuthal position of the affected loop.

![Fig. 21 Test matrix of ECC injection experiments, indication of observed flow regime [27]](image)

![Fig. 22 Time evolution of the mixing scalar at the two downcomer sensors in the experiment with 10% loop flow rate and no density difference](image)

![Fig. 23 Time evolution of the mixing scalar at the two downcomer sensors in the experiment with 10% loop flow rate and 10% density difference](image)

The maximum tracer concentration of the ECC water in the downcomer is 20% of the injected water concentration at the upper sensor and 8% at the lower sensor. Such a velocity field is typical for single-loop operation. It has its maximum at the opposite side of the downcomer and a minimum at the azimuthal position of the running loop, which has been found in velocity
measurements by means of a laser-Doppler anemometer at the ROCOM test facility [18], too.

Figure 23 shows the experiment, carried out at the same flow conditions, but the density difference between the injected ECC water and the primary loop coolant is now 10%. In that case a streak formation of the water with higher density is observed. At the upper sensor, the ECC water covers a much smaller azimuthal sector. The density difference partly suppresses the propagation of the ECC water in circumferential direction. The ECC water falls down in an almost straight streamline and reaches the lower downcomer sensor directly below the affected inlet nozzle. Only later, coolant containing ECC water appears at the opposite side of the downcomer. The maximum concentration values observed at the two downcomer sensors are significantly higher than in the case without density differences, i.e. 37.5% and 20.0% from the initial concentration in the ECC water tank. The visualizations of the behavior of the ECC water in the downcomer reveals that in case of momentum driven flow, the ECC water covers nearly the whole perimeter of the upper sensor and passes the measuring plane of the lower sensor mainly at the opposite side of the downcomer. When, on the other hand, density effects are dominating, the sector at the upper measuring device covered by the ECC water is very small. The ECC water falls down straightly and passes the sensor in the lower part of the downcomer below the inlet nozzle of the working loop.

Subsequently, variations of the density were carried out to identify the transition region between momentum driven and density driven flow. A density difference of 4% was found to belong to an intermediate region. In this case, the ECC water reaches the opposite side of the downcomer and the region below the inlet nozzle position near the lower sensor almost at the same time. That means, that one part of the ECC water follows the stream lines of the external momentum driven flow field and another part directly falls down due to the internal momentum created by density differences.

Based on these observations, the set of experiments conducted according to the matrix in figure 21, was divided into three groups: density dominated flow (○), momentum dominated flow (△) and the transition region (×). The conditions at the inlet into the downcomer were used to calculate Froude-numbers of the experiments according to the following formula [6]:

$$Fr = \frac{V_{\text{in}}}{\sqrt{g \cdot s \cdot \frac{\rho_{\text{in}} - \rho_a}{\rho_{\text{in}}}}}$$

where $V_{\text{in}}$ is the velocity at the reactor inlet (combined loop and ECC flow), $g$ is the gravitational acceleration, $s$ is the width of the downcomer, $\rho_{\text{in}}$ the density of the incoming flow, calculated with the assumption of homogeneous mixing between ECC and loop flow, and $\rho_a$ the density of the ambient water in the downcomer. Lines of constant Froude-numbers calculated by means of this formula are shown in figure 21. All experiments, identified as density dominated are located in the region left of the iso-line $Fr = 4.0$ and all momentum dominated points are found right of the iso-line $Fr = 7.0$. These two numbers are critical Froude numbers separating the two flow regimes for the ROCOM test facility. Between the two criterial values a transition region is located.

Based on these observations, the set of experiments conducted according to the matrix in figure 21, was divided into three groups: density dominated flow (○), momentum dominated flow (△) and the transition region (×). The conditions at the inlet into the downcomer were used to calculate Froude-numbers of the experiments according to the following formula [6]:

\[ Fr = \frac{V_{\text{in}}}{\sqrt{g \cdot s \cdot \frac{\rho_{\text{in}} - \rho_a}{\rho_{\text{in}}}}} \]

where $V_{\text{in}}$ is the velocity at the reactor inlet (combined loop and ECC flow), $g$ is the gravitational acceleration, $s$ is the width of the downcomer, $\rho_{\text{in}}$ the density of the incoming flow, calculated with the

Density effects are extremely developed in an experiment with stagnant coolant in the primary loop (Fig. 24), where the fluid circulation is initiated only by starting the ECC injection (injection time was 40 s). At the upper sensor, the ECC water appears unmixed and covers a sector of only about 15 degree. The data from the lower downcomer sensor show clearly buoyancy induced turbulent structures. As can be concluded from these data, the water with higher density accumulates in the lower plenum.

**X. SUMMARY**

ROCOM is a test facility for experimental coolant mixing studies over a wide variety of possible scenarios. It is equipped with advanced instrumentation, which delivers high-resolution information characterizing either temperature or boron concentration fields in
the investigated pressurized water reactor. The obtained huge data basis and the fact that the coolant mixing was studied not only by characterizing the resulting distributions at the core entrance, but also by high resolution measurements at two axial positions in the downcomer, represent unique contributions to understanding of coolant mixing in pressurized water reactors.

Experimental results show that the mixing is incomplete, and that for different scenarios, the distributions of the mixing scalar show large qualitative differences. The test results were successfully applied for CFD code validation. 3D simulations were carried out with the code CFX-4.

An approach to generalize experimentally obtained mixing scalars was developed on the basis of the superposition of system responses on Dirac impulse-like perturbations. After coupling the obtained simplified model with a three-dimensional neutron kinetics code (DYN3D) a hypothetical transient caused by pump start-up with a deborated slug in the corresponding loop was analyzed. Despite extremely conservative assumptions about the quantity of the deborated coolant (36 m$^3$) and additional unfavorable conditions, like stuck control rods etc., no fuel rod damage was theoretically predicted.

ACKNOWLEDGEMENT

The project this paper is based on is funded by the BMWi (Federal Ministry of Economics and Technology of Germany) and is registered with No. 150 1216.

NOMENCLATURE

- $a$ index ambient
- $C_B$ g/l Boron concentration
- CFD abbr. computational fluid dynamics
- ECC abbr. emergency core cooling
- $Fr$ 1 Froude number
- $g$ m/s$^2$ gravity constant
- $in$ index inlet
- $Re$ 1 Reynolds number
- RPV abbr. reactor pressure vessel
- $s$ m width of downcomer
- $t$ s time
- $T$ deg C temperature
- $x$ m coordinate
- $y$ m coordinate
- $z$ m coordinate
- $\theta$ 1 mixing scalar
- $\rho$ kg/m$^3$ density
- $\sigma$ m$^2$/s surface tension

REFERENCES


27. S. Kliem, H.-M. Prasser, G. Grunwald, U. Rohde et al. (2002), ROCOM Experiments on the influence of density differences on the coolant mixing inside the reactor pressure vessel of a PWR, Annual Meeting on Nuclear Technology ’02, accepted.