CFX SIMULATIONS OF ROCOM SLUG MIXING EXPERIMENTS

Fabio Moretti[1], Daniele Melideo[1], Francesco D'Auria[1],
Thomas Höhne, and Sören Kliem

1. Introduction

In the safety analysis of nuclear reactors, a number of scenarios have to be addressed in which a safety-relevant role is played by the space and time distribution of primary coolant physical and/or thermodynamic properties, such as temperature, density, concentration of additives etc. [1]. For example, transients have to be analyzed which are featured by a perturbation of the coolant properties at the reactor core inlet, since such a perturbation can introduce a positive reactivity and determine a rapid power excursion, potentially leading to core damage. Those transients include the so-called boron dilution scenarios in Pressurized Water Reactors (PWR), leading to a reduction in the boron concentration at the core inlet, and Main Steam Line Break (MSLB) accidents in PWRs, leading to an overcooling in the loop, and thus to relatively cold water reaching the core inlet.

Another typical problem related to the distribution of coolant properties, is whether an Emergency Core Cooling (ECC) injection following a Small Break Loss-of-Coolant Accident (SB-LOCA) may lead or not to a Pressurized Thermal Shock (PTS) scenario, due to the relatively cold injected water being not sufficiently mixed with the water already present in the cold legs.

The ANSYS CFX-10.0 CFD software has been used for simulating a number of experiments conducted on the German ROCOM test facility at Forschungszentrum Dresden-Rossendorf (FZD). One experiment in particular is addressed here, which reproduced the injection of a de-borated water slug (simulated by a tracer) into the RPV of a PWR with all circulation pumps at steady-state operation.

The results of the simulations (in terms of tracer space and time distribution) were compared against the experimental data kindly made available by FZD to DIMNP (Department of Mechanics, Nuclear and Production Engineering, University of Pisa, Italy) within a Cooperation Agreement, in order to evaluate the capabilities of the code in predicting the flow phenomena (in particular the turbulent mixing) affecting the addressed scenarios.

2. The ROCOM Test Facility

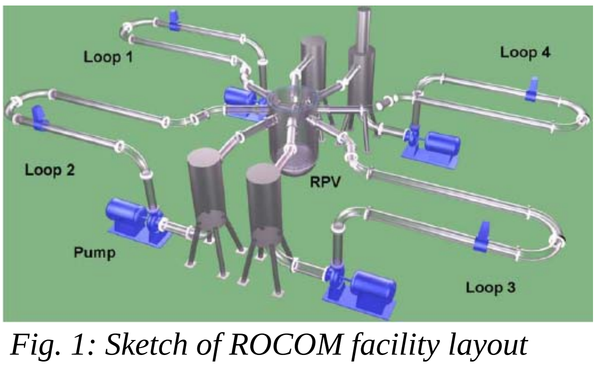

Fig. 1: Sketch of ROCOM facility layout

The ROCOM (Rossendorf Coolant Mixing Model) test facility is a 1/5 scaled model of the primary circuit of a Konvoi-type PWR reactor. It was built by FZD with the purpose of investigating the coolant mixing phenomena occurring in the reactor pressure vessel (RPV) of a PWR, and to provide experimental data for CFD code validation. A sketch of the facility layout is shown in Figure 1. Descriptions of the facility and of

[1] University of Pisa, Italy

its equipment can be found in [2], [3] and [4].

The local tracer concentration is derived from measurements of the electrical conductivity of the fluid (after obtaining proper calibration curves). The conductivity is measured by the so-called wire mesh sensors designed by the FZD experts.

When a ROCOM experiment is conducted to reproduce a boron dilution scenario, the injection of salted water is used to simulate the perturbed coolant (i.e. the coolant with low or no boron concentration), while the clear (i.e. non salted) water already present in the loops simulates the normally borated water in the real reactor. The measured values are normalized with respect to the values of conductivity that characterize the "clear water" and the "salted water", by defining a scalar quantity named the mixing scalar (MS in the following) and defined by Equation (1).

$$\theta_{x,y,z,t} = \frac{\sigma_{x,y,z,t} - \sigma_0}{\sigma_1 - \sigma_0} \cong \frac{C_{x,y,z,t} - C_0}{C_1 - C_0} \qquad (1)$$

where σ and C indicate the conductivity and the boron concentration respectively, while the sub-scripts 0 and 1 correspond to the unaffected water and the water subjected to the perturbation.

3. The Simulated Experiment

The simulated experiment is a steady-state test, and is identified as ROCOM_STAT_02. It was conducted with the four pumps running at 25 % of nominal speed (i.e. 46.25 m^3/h volume flow rate per loop), and injecting the tracer for 35 s. Figure 2 shows the time history of the cross-section averaged MS as measured at inlet nozzle sensor, along with the maximum and the minimum values.

The tracer concentration is thus not perfectly uniform over the nozzle cross section (despite the use of the mixing device), and maximum deviations are in the range -/+ 15 %. The Figure also shows a new increase in the measured MS after 45 s from the beginning of the experiment: this is because the injected tracer has gone through the whole circuit and enters the vessel for the second time.

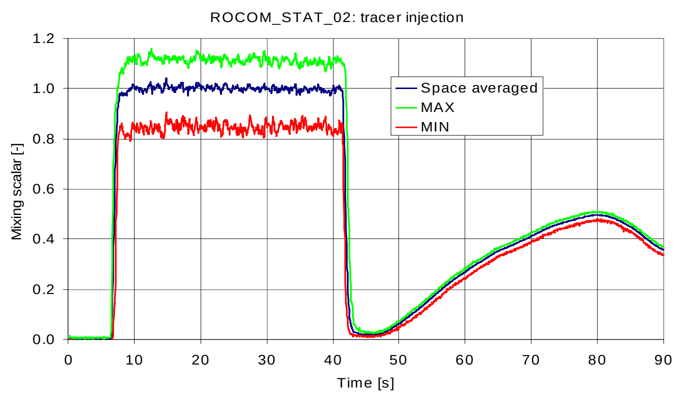

Fig. 2 ROCOM_STAT_02 experiment: time history of the inlet MS

Five realizations were performed for these experiments, and the time histories of the measured MS at each measuring position were averaged, so as to filter the effect of the turbulent fluctuations. The maximum reference conductivity value needed for the non-dimensionalisation (i.e. to calculate the MS) is derived from the space-averaged plateau-averaged value at inlet nozzle sensor (for the steady-state experiments), or from the maximum in time space-averaged value at inlet nozzle sensor (for the transient experiments). The available experimental data include the time-dependent MS at each measuring point of each sensor, with a time step of 0.05 s (i.e. 20 Hz frequency).

4. CFD simulations

The meshing tool used was ANSYS ICEM 10.0 [5]. Since the considered experiments and the related CFD calculations focus on the mixing phenomena occurring inside the reactor vessel, whereas no attention is paid to the flow phenomena in the other parts of the Reactor Coolant System (RCS) such as cold and hot legs, circulation pumps, etc.), the selected computational domain includes only the following parts of the ROCOM facility (Figure 3):

- downcomer (DC) (including 4 inlet nozzles)
- lower plenum (LP)
- core simulator (CS)
- upper plenum (UP)

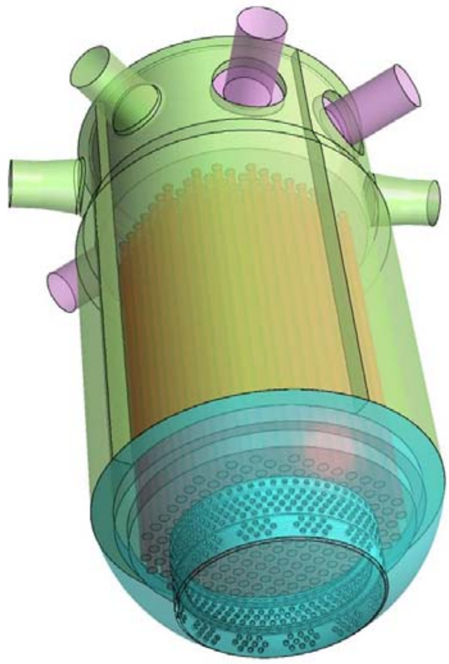

Fig. 3: ROCOM computational domain

The DC was modelled according to the real geometry of ROCOM without any simplification. In particular, the diameter variations both in the inlet nozzle and in the DC were taken into account, as well as the fillet radius in the connection between nozzle and DC, which was shown by preliminary CFD studies to sensibly affect the mass flow distribution in the DC. The LP sub-domain is defined by: the interface with DC part, the inner surface of the vessel bottom, the side and lower surfaces of the support plate, and the boundaries of the perforated drum. The drum was modelled along with its 410 holes (15 mm diameter), as shown in Figure 4.

The CS consists in 193 tubes connecting the LP to the UP. Its length has been extended downward so as to "replace" the complex geometry of the lower support plate with a simplified, tube-based geometry. This geometry simplification obviously affects the pressure losses that the flow encounters when crossing the plate. This was taken into account in the CFD simulations by defining additional pressure losses.

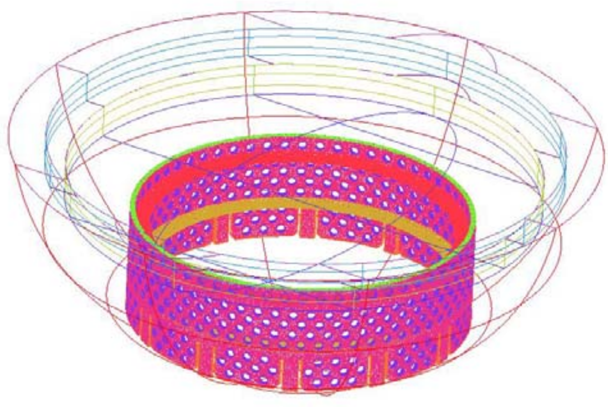

Fig. 4: LP solid model (perforated drum)

The UP has a straightforward geometry, since it is simply made of five cylinders (the plenum itself and the four outlet nozzles) and does not have any internals.

Three of the several grids created were used for the simulations described in this paper. In all cases, around one half of the whole number of cells is present in the LP region, due to the presence of smaller scale geometric details. The DC grid was obtained using either hexa-meshing (A01, see Figure 5) or tetra-meshing (A04, A07). In all cases the LP was meshed with tetrahedra (Figure 6), because a hexa-meshing would be too difficult to achieve due to the geometric complexity. The LP of grid A07 has layers of prism elements to better comply with turbulent treatment at the walls. The CS region of grids A01 and A04 was obtained through axial extrusion of 2D meshes (triangles and quadrangles) of the tubes cross sections, thus generating prisms and hexahedra, and the UP was meshed with hexahedra. In grid A07 the CS is replaced by a "reduced core" meshed with prisms (extrusion of triangles), while a cylindrical outlet volume, meshed with tetrahedral, is present instead of the UP.

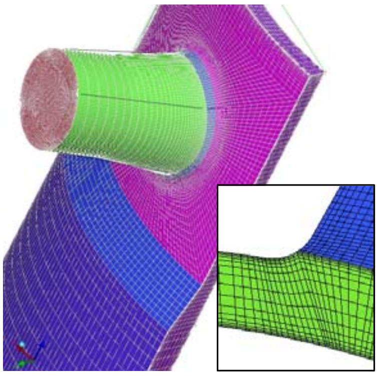

Fig. 5: DC mesh (A01 grid)

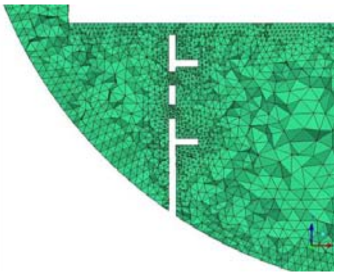

Fig 6: LP mesh (A01, A04)

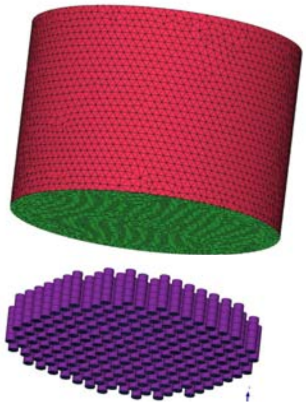

Fig. 7: Reduced core (A07 grid)

The simulations were performed with the ANSYS CFX-10.0 package [6]. All the calculations performed share the following features (see also Table 1):

- Working fluid: incompressible water at 0.98 bar, 25°C
- Density: 997 kg/m^3
- Dynamic viscosity: 8.899 x 10-4 kg m^{-1} s^{-1}
- Constant inlet velocity
- Uniform inlet velocity profile (0.73 m/s, corresponding to 25 % of nominal volume flow rate)
- Pressure-controlled outlet boundary condition
- Uniform inlet turbulent intensity profile (either 5 % or 10 %)
- MS injected into loop #1 inlet nozzle (non-uniform extracted from sensor data)
- Upwind discretization scheme for advection terms
- k-ε or SST turbulence model

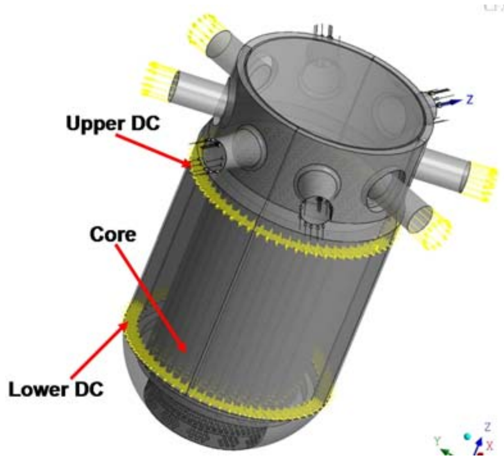

Fig. 8: Locations of monitor points

The purpose of the simulations was to calculate the local and instantaneous value of the MS, and then to compare it against the available experimental data. The following monitor points (also shown in Figure 8) were defined for the MS, according to the geometric location and configuration of the wire mesh sensors:

- 256 upper DC monitor points located in the upper part of the DC (4 radial and 64 azimuthal measuring positions)
- 256 lower DC monitor points located in the lower part of the DC (4 radial and 64 azimuthal measuring positions)

Table 1: Summary of CFX calculations for ROCOM_STAT_02 test

Run #	1	2	3	4	5
Run name	**A01_Stat2_case1**	**A01_Stat2_case2**	**A01_Stat2_case3**	**A04_Stat2_case3**	**A07_Stat2_case1**
Computational domain	whole RPV				with “reduced core”
Turbulence model	k-ε				SST
Restart from	-	A01_Stat2_case1	A01_Stat2_case2	A01_Stat2_case2 (with interpolation)	-
MS injection	Continuous				Slug
Inlet MS profile	Uniform	Non-uniform (with user function)			
Inlet turbulent intensity	5 %		10 %	5 %	
Equations solved	All	MS only	All		

5. Results

Since the addressed experiment involves symmetric and constant pumps operation, the resulting flow field in the RPV consists of four symmetric “flow sectors”, each one corresponding to one loop. The flow entering the RPV from the four inlet nozzles, first impinges against the barrel outer wall, so that complex local flow patterns develop in the inlet region. However the flow in the downcomer below the diffusion zone (diameter variation) is mainly directed downwards, as shown by the streamlines in Figure 9.

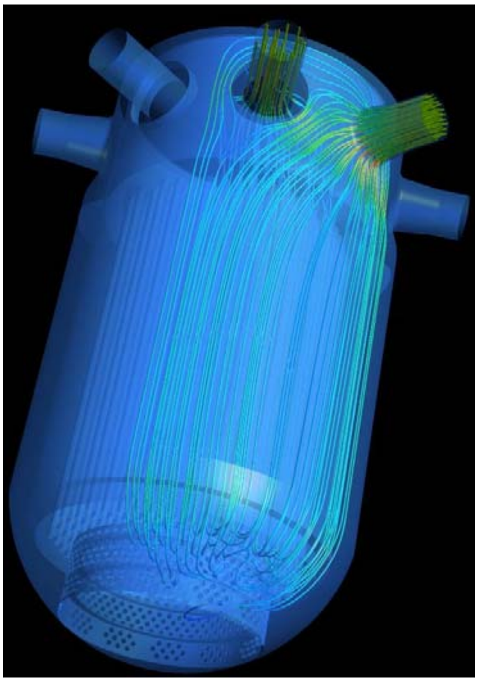

Fig. 9: CFD results: streamlines from loop #1

The downcomer flow is affected by turbulent mixing, which causes the MS field to diffuse in the transverse direction (i.e. azimuthally). This is shown by the experimental data represented in Figure 10 and Figure 11. In particular, the Figure 10 shows the comparison of MS azimuthal profile in the upper part of the downcomer: the agreement between the experiment and the calculations is good (max. 5 % difference of MS at local positions). The agreement between numerical predictions and experiment becomes poorer in the lower part of the downcomer, as evident from Figure 11. The MS field experiences a relatively strong diffusion in the azimuthal direction, which “smears” the profile, while the spatial gradients predicted by the simulations keep steeper. As a consequence, the predicted maximum values are higher than the experimental ones, and the MS perturbation does not affect the two adjacent sectors.

What stated above suggests that the code, despite the use of a first-order discretization scheme (i.e. “upwind”) for the advection terms, yet underestimates the turbulent diffusivity (in other words, the effectiveness of turbulent mixing). The best results are those yielded by calculation #4: this can be explained with the use of tetrahedral elements in the DC grid (instead of hexahedra), which introduce higher numerical diffusion and thus partly compensate for the above mentioned underestimation.

From these results about the mixing in the DC, one can expect that the predicted MS perturbation will affect a smaller number of fuel element positions than in the experiment, and that MS values in the most affected fuel element positions will be higher.

6. Summary and conclusions

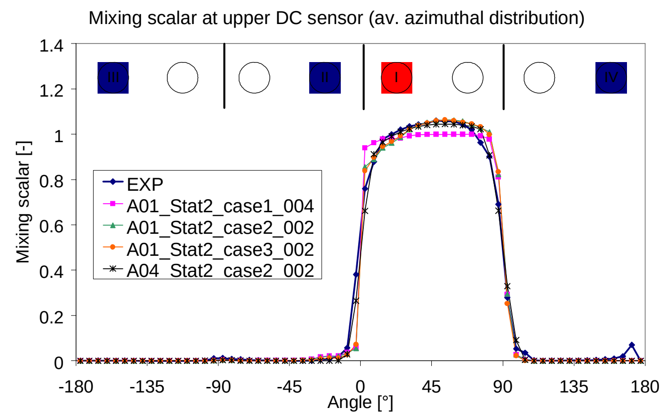

Fig. 10: Azimuthal profile of MS at the upper DC sensor

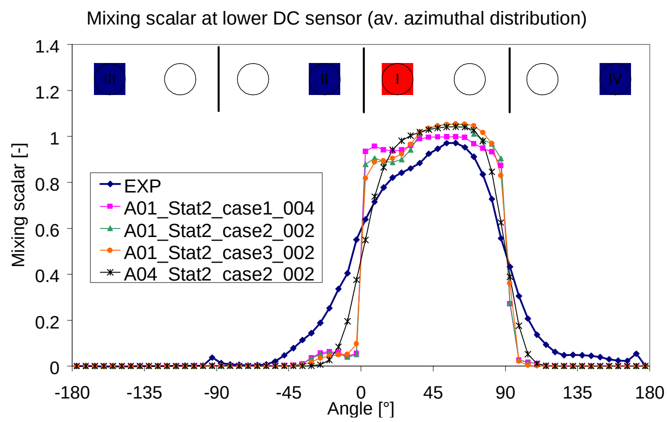

Fig. 11: Azimuthal profile of MS at the lower DC sensor

The CFD code CFX has been used, in the framework of CFD code validation activities ongoing at the University of Pisa, for the simulation of a slug-mixing experiment, conducted on the ROCOM test facility at Forschungszentrum Dresden-Rossendorf (FZD). Much effort was spent in obtaining several fine and high-quality computational grids, which could cope both with the relatively high geometric complexity of ROCOM vessel internals, and with computing power limitations as well. Several parallel simulations were set-up and run, based on different meshing solutions, numerical options and modelling assumptions. Then the numerical results, were compared against the experimental data, in terms of tracer concentration space and time profiles both in the downcomer and at the core inlet. The CFD simulations resulted to be able to correctly predict the formation of the perturbed region in the downcomer and in the lower plenum. Quantitatively, the code tends to predict steeper gradients of the concentration space profiles than in the experiment, that is the effectiveness of the turbulent mixing is generally under-predicted. This may be related to the limitations of 2-equation turbulence models in deal with the high anisotropy of the turbulent structures forming in the downcomer. This aspect was not further investigated.

Although higher accuracy in prediction of turbulent mixing would certainly be desirable for “best-estimate” purposes, the “conservativeness” of the results obtained should be put in evidence, since they predict a less mitigating effect played by the turbulence mixing in the assumed accidental scenarios.

References

[1] Moretti, F., Del Nevo, A., D’Auria, F., 2006, Asymmetrical Boron Concentration / Temperature Events: Review of US Position and Approaches – Review of EU Framework Programmes Related to Computational Tools, DIMNP NT 575 (06), Pisa.

[2] Grunwald, G., Höhne, T., Kliem, S., Prasser, Richter, K.-H., H.-M., Rohde, U., Weiss, F.-P., 2002, Versuchsanlage ROCOM zur Untersuchung der Kühlmittelvermischung in Druckwasserreaktoren - Ergebnisse quasistationärer Vermischungsexperimente. FZR Report 348.
[3] Höhne, T., Kliem, Prasser, H.-M., Rohde, U. , 2003, Experimental and numerical studies inside a reactor pressure vessel. 4th ASME/JSME Joint Fluids Engineering Conference, Hawaii, USA. CD-ROM.
[4] Rohde, U. et al., 2004, Description of the slug mixing and buoyancy related experiments at the different test facilities, EU/FP5 FLOMIX-R Project, FLOMIX-R-D09, Germany.
[5] ANSYS Inc., 2005a, ANSYS ICEM-CFD 10.0 User Manual, (embedded in the software package)
[6] ANSYS Inc., 2005b, ANSYS CFX-10.0 User Manual, (embedded in the software package)