Fluid mixing and flow distribution in a primary circuit of a nuclear pressurized water reactor—Validation of CFD codes

U. Rohde a,*, T. Höhne a, S. Kliem a, B. Hemström b, M. Scheuerer c, T. Toppila d, A. Aszodi e, I. Boros e, I. Farkas f, P. Mühlbauer g, L. Vyskocil g, J. Klepac h, J. Remis h, T. Dury i

a Forschungszentrum Rossendorf, Dresden (DE), Germany
b Vattenfall Utveckling AB, Altvkarleby (SE), Sweden
c Gesellschaft für Anlagen-und Reaktorsicherheit, Garching (DE), Germany
d Fortum Nuclear Services, Espoo (FIN), Finland
e Budapest University of Economics and Technology (HU), Hungary
f Atomic Energy Research Institute AERI, Budapest (HU), Hungary
g Nuclear Research Institute, Rez (CZ), Czech Republic
h VUJE Trnava (SK), Slovakia
i PSI Villigen (CH), Switzerland

Received 6 July 2006; received in revised form 5 March 2007; accepted 5 March 2007

Abstract

The EU project FLOMIX-R was aimed at describing the mixing phenomena relevant for both safety analysis, particularly in steam line break and boron dilution scenarios, and mixing phenomena of interest for economical operation and the structural integrity. This report will focus on the computational fluid dynamics (CFD) code validation. Best practice guidelines (BPG) were applied in all CFD work when choosing computational grid, time step, turbulence models, modelling of internal geometry, boundary conditions, numerical schemes and convergence criteria. The strategy of code validation based on the BPG and a matrix of CFD code validation calculations have been elaborated. CFD calculations have been accomplished for selected experiments with two different CFD codes (CFX, FLUENT). The matrix of benchmark cases contains slug mixing tests simulating the start-up of the first main circulation pump which have been performed with three 1:5 scaled facilities: the Rossendorf coolant mixing model ROCOM, the Vattenfall test facility and a metal mock-up of a VVER-1000 type reactor. Before studying mixing in transients, ROCOM test cases with steady-state flow conditions were considered. Considering buoyancy driven mixing, experimental results on mixing of fluids with density differences obtained at ROCOM and the FORTUM PTS test facility were compared with calculations. Methods for a quantitative comparison between the calculated and measured mixing scalar distributions have been elaborated and applied. Based on the “best practice CFD solutions”, conclusions on the applicability of CFD for turbulent mixing problems in PWR were drawn and recommendations on CFD modelling were given. The results of the CFD calculations are mostly in-between the uncertainty bands of the experiments. Although no fully grid-independent numerical solutions could be obtained, it can be concluded about the suitability of applying CFD methods in engineering applications for turbulent mixing in nuclear reactors.

© 2007 Elsevier B.V. All rights reserved.

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 pressurized 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, steam produced in the reactor core is condensed in the steam generator, and a slug of low boron condensate will accumulate at the cold leg of the primary circuit. During start-up of coolant circulation after refilling the primary circuit with emergency cooling (ECC) water or by switching on

* Corresponding author. Tel.: +49 351 260 3460; fax: +49 351 260 2383.
E-mail address: rohde@fz-rossendorf.de (U. Rohde).
the first main coolant pump (MCP), this slug will be transported into the reactor core causing a significant reactivity insertion by decreasing the amount of neutron absorber. The mixing of the deborated condensate with borated water in the reactor pressure vessel is in that case the only mitigative mechanism to prevent severe accident consequences (Kliem et al., 2004a,b). The mixing is also relevant in overcooling transients, when the coolant temperature in one or more loops decreases, e.g. due to a leak in the secondary side steam system (Kliem et al., 1999). A strong decrease of the coolant temperature does also cause a reactivity insertion due to the enhanced moderation of neutrons.

Mixing is relevant not only for nuclear safety, but also for structural integrity. In the case of LOCAs, 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 can be dangerous for the RPV integrity. Mixing is even of relevance for normal reactor operation, e.g. for determination of the coolant temperature distribution at the core inlet in the case of partially switched off MCPs.

In the EC project FLOMIX-R, slug mixing and flow distribution in the RPV has been comprehensively investigated experimentally and simulated by using computational fluid dynamics (CFD) tools. Within this project, a comprehensive data base on coolant mixing inside the RPV was created (Rohde et al., 2005). Slug mixing experiments have been performed with several 1:5 scaled facilities representing different European reactor types: the Rossendorf coolant mixing model ROCOM and the Vattenfall test facility, modelling a German Konvoi type and a Westinghouse type three-loop PWR, respectively (Alavyoon et al., 2004, a,b). Additional data on slug mixing in a VVER-1000 type reactor gained at a 1:5 scaled metal mock-up at EDO Gidropress are provided (Bezrukov et al., 2002). Experimental results on mixing of fluids with density differences obtained at ROCOM and the FORTUM PTS test facility are made available (Tuomisto, 1987; Kliem et al., 2004a,b, Rohde et al., 2005). Data from steady-state mixing experiments at ROCOM (Kliem et al., 2004a,b) and plant commissioning test data from the NPP Paks and Loviisa (Elter, 2002; Tsimbalov et al., 1982) were gained and used to contribute to the validation of CFD codes for the analysis of turbulent mixing problems. The commercial CFD codes CFX4, CFX5 and FLUENT6 have been used (CFX, 2001, 2003; FLUENT, 2003).

Table 1

<table>
<thead>
<tr>
<th>Run ROCOM_01</th>
<th>Flow rate (m³/h)</th>
<th>Loop 1</th>
<th>Loop 2</th>
<th>Loop 3</th>
<th>Loop 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>01</td>
<td>185</td>
<td>185</td>
<td>185</td>
<td>185</td>
<td>185</td>
</tr>
<tr>
<td>04</td>
<td>185</td>
<td>185</td>
<td>185</td>
<td>185</td>
<td>Back flow</td>
</tr>
<tr>
<td>08</td>
<td>203.5</td>
<td>166.5</td>
<td>185</td>
<td>185</td>
<td>185</td>
</tr>
<tr>
<td>09</td>
<td>222</td>
<td>148</td>
<td>185</td>
<td>185</td>
<td>185</td>
</tr>
</tbody>
</table>

The loop with the tracer solution injection in the test matrix is the loop number one.

Tables 1–3 show the test matrices for code validation.

In the steady-state mixing tests at ROCOM (Table 1), besides the symmetrical case ROCOM_01, there were considered experiments with one non-working loop (ROCOM_004) and with asymmetrical mass flow rate distribution among the loops (ROCOM_008, _stat09). In the ROCOM slug mixing tests (see Table 2), the tests ROCOM_01, _02 and _03 correspond to the pump start-up curve in the real plant, while the slug size was varied. In the test ROCOM_08, the mass flow ramp was varied according to Froude scaling. One slug mixing test from the Vattenfall facility and two from the Gidropress test rig were selected for calculations, too. Concerning buoyancy driven mixing cases, three tests from the Fortum PTS facility were calculated with different combinations of mass flow rates Q_inj in the ECC injection loop and Q_A and Q_B in the loops neighbouring to the injection loop (see Table 3). Two ROCOM tests with different injection loop flow rate and density differences were selected for the calculations.

The main objective was to investigate how well mixing during boron dilution transients in PWRs can be modelled by CFD codes. The competitiveness of CFD is continuously growing due to the rapid developments in computer technology. However, computer capacity is still, and will be for a foreseeable future, a

Table 2

<table>
<thead>
<tr>
<th>Run</th>
<th>Ramp length (s)</th>
<th>Final volume flow rate (m³/h)</th>
<th>Slug volume (m³)</th>
<th>Initial slug position (m)</th>
<th>Reactor type</th>
</tr>
</thead>
<tbody>
<tr>
<td>ROCOM_01</td>
<td>14</td>
<td>185.0</td>
<td>40.0</td>
<td>10.0</td>
<td>PWR Konvoi 4-loop</td>
</tr>
<tr>
<td>ROCOM_02</td>
<td>14</td>
<td>185.0</td>
<td>20.0</td>
<td>10.0</td>
<td>PWR Konvoi</td>
</tr>
<tr>
<td>ROCOM_03</td>
<td>14</td>
<td>185.0</td>
<td>4.0</td>
<td>10.0</td>
<td>PWR Konvoi</td>
</tr>
<tr>
<td>ROCOM_08</td>
<td>28</td>
<td>92.5</td>
<td>4.0</td>
<td>10.0</td>
<td>PWR Konvoi</td>
</tr>
<tr>
<td>VATT-02</td>
<td>16</td>
<td>429</td>
<td>8.0</td>
<td>10.0</td>
<td>Westinghouse 3-loop</td>
</tr>
<tr>
<td>GP-1</td>
<td>2</td>
<td>175</td>
<td>8.5</td>
<td>14.5</td>
<td>VVER-1000</td>
</tr>
<tr>
<td>GP-2</td>
<td>8</td>
<td>470</td>
<td>8.5</td>
<td>14.5</td>
<td>VVER-1000</td>
</tr>
</tbody>
</table>

*Related to the original reactor.

Table 3

<table>
<thead>
<tr>
<th>Nr</th>
<th>Q_A (l/s)</th>
<th>Q_B (l/s)</th>
<th>Q_HPI (l/s)</th>
<th>Δρ_HPI/ρ</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fortum PTS tests</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>2.31</td>
<td>1.87</td>
<td>2.31</td>
<td>0.16</td>
</tr>
<tr>
<td>20</td>
<td>0</td>
<td>2.31</td>
<td>1.87</td>
<td>0.16</td>
</tr>
<tr>
<td>21</td>
<td>1.87</td>
<td>1.87</td>
<td>1.87</td>
<td>0.16</td>
</tr>
<tr>
<td>ROCOM buoyancy mixing tests</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>D10M05</td>
<td>2.72</td>
<td>1.00</td>
<td>–</td>
<td>0.10</td>
</tr>
<tr>
<td>D05M00</td>
<td>0</td>
<td>1.00</td>
<td>–</td>
<td>0.05</td>
</tr>
</tbody>
</table>
limiting factor for the capacity for CFD calculations to produce completely accurate results. Simplified models for describing turbulence therefore have to be used and the computer capacity put restrictions on the resolution in space and time that one can use in a CFD calculation. This leads to modelling errors and numerical errors that give more or less inaccurate results. Validation of the quality and trust of different approaches in CFD calculations are therefore needed.

2. Application of the best practice guidelines for CFD

The CFD code validation was focussed mainly on a number of benchmark cases from the steady-state mixing experiments, slug mixing tests and experiments with density differences. So-called best practice guidelines (BPG) were used for quality assurance of the validation calculations. ERCOFTAC BPG (Casey and Wintergerste, 2000), which have been specified for nuclear reactor safety calculations within the ECORA project (Menter et al., 2002) were applied. 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. The separation of numerical errors from model errors allows then valid conclusions on model performance. Numerical errors are minimized by optimising the computational mesh, numerical schemes, convergence criteria and time step. Another kind of errors are uncertainties rising from insufficient information about the problem definition and set-up. These uncertainties can be quantified by performing calculations with variations of the unknown parameters and by a subsequent analysis on the influence of these parameters. This belongs to boundary positions, boundary conditions and internal geometry modelling. Turbulence models are the most relevant physical models responsible for model errors. Only if sensitivity tests are made the solution errors and model errors can be quantified and only then you can get an indication on how good your CFD calculations are. Sensitivity tests for the following aspects were considered:

- Grid size.
- Convergence criteria.
- Round-off error.
- Time step size.
- Turbulence model.
- Inlet boundary position and inlet boundary condition.

Fig. 1 shows examples for computational grid and time step optimisation using the CFX-5 code. The number of mesh points for the discretisation of the solution area has to be increased until convergence of the solution has been achieved (mesh with about 750,000 elements using the Upwind Discretisation Scheme UDS). The same procedure has to be applied concerning time step refinement (see also Fig. 1).

However, in practical applications with complicated geometry and complex flow phenomena like the mixing problems considered, a really grid and time step independent solution can often not be reached by practical reasons (computer resources). In these cases, so-called production meshes were used in the calculations. The production mesh is an optimum between maximum possible refined grid on the one hand, but with omitting parts of the flow domain, which were found to be of small impact on the results, e.g. the cold leg loops, and neglecting or simplifying some geometrical details, e.g. by modelling complicated structures as porous body domain. The production mesh is not yet a mesh, for which grid-independent solution was reached. In general, the choice of the production mesh is dependent from the process to be simulated. The production mesh for momentum driven mixing calculations might be different from that one
for the analysis of buoyancy controlled mixing. The production mesh for the CFX-5 calculations, an unstructured grid with about 7 million elements, is shown in Fig. 2.

Concerning numerical solution schemes, higher order schemes are preferred due to higher accuracy. Fig. 3 shows the velocity distribution (vertical component) in the core inlet plane calculated with the FLUENT code for steady-state conditions with one running pump in the Vattenfall facility by applying first order and second order scheme. Using the first order scheme, the solution is well converging, but physically wrong. The position of the maximum velocity observed in the experiment is not met in the first order solution. The reason is to be seen in the influence of inlet boundary conditions. The bend in the cold leg pipe not far from the RPV inlet nozzle causes an asymmetric velocity profile and turbulence parameters distribution at the RPV inlet. This impact of the bend is not well reproduced in the more diffusive first order scheme calculation. The disturbance caused by the bend is not “transported” to the core inlet plane, the diffusive scheme “forgets” about it.

3. Post-test calculations of ROCOM steady-state mixing tests

Fig. 4 shows a comparison between CFD solutions and experiment for the steady-state mixing test ROCOM_stat01 with four running pumps. The sector formation and the high maximum mixing scalar values are well reproduced in the calculation. The best agreement was achieved, when a transient calculation for the steady-state conditions was performed, because the velocity field in the flow domain is not really time-independent even with steady-state boundary conditions. Macroscopic fluctuations occur, which are averaged from the transient solution.

Parallel to CFX, the CFD code FLUENT has been used for performing second code calculations for the basic steady-state mixing test ROCOM_stat01 and for calculating additional steady-state tests with asymmetric operation of loops (ROCOM_stat04, ROCOM_stat09). Second code calculations are an important contribution with respect to different mesh structures, specific turbulence models and particularly for modelling of internal structures. A comparison of the different FLUENT and CFX-5 results with measurement data (plateau averaged mixing scalar distribution at core inlet) for ROCOM_stat01 is also shown in Fig. 4. The results of the calculations for the additional tests ROCOM_stat04 and ROCOM_stat09 are described in Hemström et al. (2005).

The velocity field in the downcomer shows a qualitatively good agreement between the CFX-4 and CFX-5 calculations and the experimental results (LDA measurements at ROCOM (Prasser et al., 2003)). Particularly, the calculations confirm the location of minimum flow velocities below the inlet nozzles found in earlier experiments (Ulrych and Weber, 1983). A maximum velocity exists at azimuthal positions between the two inlet, respectively, the two outlet nozzles. A comparison
The calculated and measured velocity distributions agree mostly within the uncertainty band of the measurements, with exception of the velocity minimum at 45° (loop 1). There is obviously a deviation from the symmetry to be expected in the measured velocity field due to equal mass flow rates in all loops. A measurement error of this magnitude in the mass flow rates can be excluded. However, in loop 1 the mixing device for the tracer injection was installed. The mixer creates an additional flow resistance, which was taken into account by adjustment of the pump head in this loop. It is assumed, that the mixer affects the velocity profile, distribution of turbulence or occurrence of swirls induced by the pumps in the corresponding cold leg pipe, what may have an impact on the velocity distribution. However, this assumption could not be checked because velocity measurements near the RPV inlet were not possible.

In general, the velocity distribution in the downcomer and mixing scalar distributions at the core inlet are qualitatively well predicted in the CFD calculations. The velocity field in the downcomer has inhomogeneous character with maximum downwards flow components in the regions in-between the inlet nozzles. A clear sector formation of the flow in the downcomer is seen. This leads to maximum mixing scalar values at the core inlet of 92–99%. That means a part of the fluid remains almost unmixed. The re-distribution of the velocity field and mixing scalar distribution in the case of asymmetric flow conditions is also qualitatively well reproduced in the calculations. However,
the mixing along the flow path in the downcomer is underpredicted in all calculations. Disturbance in the inlet boundary conditions has significant impact on the flow pattern. This can be seen in experiment ROCOM_stat01, where a disturbance in turbulence or swirl intensity caused by the mixer is assumed to be responsible for an observed perturbation in the velocity distribution.

Finer grids in the CFD simulation tend to give better results. Also modelling of perforated sheets (such as the drum in the downcomer) as real structure rather than porous medium improves quality of results. Influence of porous medium as a substitute of a perforated sheet can be, in some extent, controlled by proper definition of direction-dependent resistance of the porous medium. Other investigated effects (turbulence model, wall function, position of outlet boundary) do not have an unambiguous influence on results.

4. CFD calculations for non-buoyant slug mixing tests

4.1. The ROCOM slug mixing tests

The CFD calculations for ROCOM slug mixing tests were carried out with the CFD codes CFX-4, CFX-5 and FLUENT. The transient slug mixing cases ROCOM_01, ROCOM_02, ROCOM_03 and ROCOM_08 were calculated. In this paper, focus will be put onto CFX calculations for the benchmark case ROCOM_02. A second code calculation was performed for this case with FLUENT. A detailed description of all the calculations and their results is given in Hemström et al. (2005).

Based on the meshing studies, finally two grid types (production meshes) were used for the basic calculations: a tetrahedral production mesh (about 7 millions elements) and a hybrid production mesh (4 million elements). In the CFX-5 production meshes, all internals were modelled in detail. No porous body approach was applied. All the 194 orifices in the core support plate were modelled. The perforated drum in the lower plenum contains 410 orifices of 15 mm diameter. Inlet boundary position is at the inlet nozzle, outlet boundary at half height of the core. Both production meshes are suitable for the post-test calculation of basically all ROCOM experiments, with slightly preferences of the tetrahedral production mesh for the steady-state mixing experiments and the hybrid production mesh for the slug mixing and density driven experiments. Because no full grid independence was achieved, there are differences in the results obtained with different mesh types.

Fig. 6 shows streamlines representing the velocity field in the downcomer and lower plenum (including the perforated drum and lower support plates as porous media) at the pump start-up scenario calculated with CFX-4. Due to a strongly momentum driven flow at the inlet nozzle the horizontal part of the flow dominates in the downcomer. The injection is distributed into two main jets, the so-called butterfly distribution. In addition, several secondary flows are seen in various parts of the downcomer. Especially strong vortices occur in the areas below the non-operating loop nozzles and also below the injection loop.

Fig. 7 shows a quantitative comparison between CFD solutions and experiment for the slug mixing test ROCOM_02. The time behaviour of the maximum mixing scalar value at the core inlet is shown. The CFX-4 and CFX-5 solutions are compared with the measurement. The statistical error of the measurement data (error band P2 of two standard deviations) is shown obtained from the averaging of the data from five repetitions of the same experiment. This statistical error is not the measurement error, but is caused by macroscopic fluctuations in the flow. The CFD solutions are not always within the uncertainty band of the measurements, but the relevant values of maximum mixing scalar (safety relevant minimum boron concentration) are close together.
Sensitivity studies have shown, that the SST turbulence model and the automatic wall functions together with higher order discretisation schemes should be used if possible.

4.2. CFD calculations for Vattenfall experiments

CFD calculations have been performed for the VATT-02 test case (see Table 1). It is a slug mixing transient, with a slug with low boron concentration initially present in the cold leg pipe. Buoyancy forces are negligible. The Vattenfall experiments are described in detail in Alavyoon et al., 1995. Calculations have been made by using the CFD codes CFX-5 and FLUENT with two different grids, “Grid 1” with 200,726 cells and “Grid 2” with 1,605,808 cells. Grid 1 is a quite coarse grid and is considered to have about the minimum number of cells required to get a fair resolution of the flow field. Grid 2 is a complete refinement of Grid 1, i.e, to get Grid 2 all cells in Grid 1 are split up into eight cells. However, to be able to check if we are close to get a grid-independent solution, one would have to make a calculation with a further refined grid, ideally by a factor of eight. A calculation with such a grid with about 13 million cells was not possible with the available computer resources. Thirteen different turbulence models were tested, six versions of Reynolds Stress Models (RSM) included:

- RNG k-ε, Standard k-ε, Standard k-ε (Kato-Lauder) and Realizable k-ε.
- Standard k-ω and SST k-ω.
- RSM, with the following versions: LRR-IP, QI, SSG, Omega and Omega (BSL).

The mean and the minimum dimensionless boron concentrations (corresponding to unity minus mixing scalar) are quite well met in the FLUENT calculations for the VATT-02 slug mixing test. Fig. 8 shows mean and minimum (independently on position) dimensionless boron concentrations at the core inlet as a function of time. The calculated minimum boron concentration for the Grid 2 calculation with the RNG k-ε model is very close to the measured value (see Fig. 8, bottom). The calculated concentration is, however, delayed around 0.9 s compared to the measured concentration. This can to a large extent be due to an inaccuracy in the measured flow rate.

However, not only the minimum boron concentration should be met well, but also the positions of low boron concentrations must be captured. Two “islands” of low boron concentrations are present in both the measurement and in the CFD calculation. However, there is a displacement of the calculated minimum with respect to the measured minimum. This displacement of the concentration field is mainly a rotational displacement.

In Fig. 9, comparisons of measured and calculated velocity components are presented. The angular position of angle = 0° is defined as the angular position where the main inlet pipe enters...
the downcomer, which is to the right in the figures. Positive angles are to the right of this position, negative angles are to the left, when looking at the model from the outside standing at angle = 0°. Tangential velocity (see Fig. 9b) is defined as being positive when the flow is directed to the right when looking from an upright position outside the model, i.e. counter-clockwise. Vertical velocity (see Fig. 9a) is positive when directed upwards. Only the radial averages are shown in the figures for each angular position. One can see that the qualitative agreement is good. The important non-symmetry in the flow field found in the experiments is captured quite well.

Reasons for not getting a better agreement with measurements, particularly for the concentration distribution at the core inlet, can be found among the following items:

- Too few computational cells. Due to limited computer resources it could not be shown that a really grid-independent solution was achieved.
- A too long time step might have been used. Due to limited computer resources time step sensitivity tests have not been made.
- Simplifications in geometry can be more important than expected. Chamfers, vertical cylinders and the lowest structure in the lower plenum are neglected. Even if these structures are small they can have a significant influence on the flow field and the mixing. A sensitivity test showed that it is very important to model the horizontal structures in the lower plenum.
- Sensitivity tests showed big differences in results for different turbulence models, which indicates that more advanced turbulence models will give a better agreement with measurements.

4.3. CFD calculations for the Gidropress experiments

NRI simulated two from three slug mixing experiments provided by EDO Gidropress (Logvinov et al., 2000; Bezrukov et al., 2002; Rohde et al., 2005). The experiments simulated slug mixing in the reactor vessel of a VVER-1000 reactor. In the tests, the experimental facility was filled with hot water (temperature of 71 °C in test 1, 73.3 °C in test 2) and slug of cold water (temperature of 25.8 °C in test 1, 27.4 °C in test 2) was situated in the loop seal. The loop flow rate was then increased up to 175 m³/h (test 1) and 470 m³/h (test 2).

In test no. 1 the elliptical perforated bottom with original 1324 holes (diameter of 8 mm) was modelled as porous media zone in the first version of the grid (variant 1A), and the number of holes was decreased to 312 holes with diameter of 16 mm in the second grid (variant 1B). In test no. 2 the elliptical perforated bottom was modelled with the reduced number of holes. View of the grid and calculation domain used in test 1 is shown in Fig. 10.

In test no. 1 the elliptical perforated bottom with original 1324 holes (diameter of 8 mm) was modelled as porous media zone in the first version of the grid (variant 1A), and the number of holes was decreased to 312 holes with diameter of 16 mm in the second grid (variant 1B). In test no. 2 the elliptical perforated bottom was modelled with the reduced number of holes. View of the grid and calculation domain used in test 1 is shown in Fig. 10.

At walls, adiabatic (Neumann) boundary condition was used for the energy equation. For test 2, also adiabatic and constant temperature boundary conditions were tested (variants 2A and 2B). Realizable k-ε model was used with differential viscosity option and standard wall functions. Full buoyancy effects including the effect of buoyancy on ε were considered.

Fig. 10. Computational domain with grid used in Gidropress test no. 1—overall view.

Fig. 11 shows the dimensionless averaged temperature at the core inlet. Calculated minimum of average core inlet temperature is lower than the experimental one. The calculated average values are almost independent from the grid option (porous body or reduced number of holes). Grid with perforated bottom (variant 1B) provides better mixing than grid with porous zone (variant 1A). The underestimation of the minimum value in the calculation is an indication, that heat exchange between cold slug and walls cannot be neglected.

Calculated cold slug lags behind the experimental one. The reason for this time shift could not be clarified finally.

The transient of the average dimensionless temperature at the core inlet for the test 2 is shown in Fig. 12. In variant 2A zero wall heat fluxes are assumed as in the simulations of test 1. In variant 2B, constant wall temperature, equal to temperature of the hot water outside the cold slug, is assumed. In both cases, detailed model of the perforated bottom and realizable k-epsilon model are used. In case 2A, no heat release from the wall to the fluid is considered. The minimum temperature is underestimated in the calculation. In the case of constant wall temperature, the heat release is overestimated, because in reality, the wall will be cooled down somewhat, which leads to a reduction of the heat flux. Consequently, the minimum temperature is overestimated in the calculation. In the case of realistic heat transfer modelling...
For the assessment of agreement between calculation and measurement, not only visual and qualitative comparisons have to be performed, but also a quantitative estimation of the deviations. Deviations between calculation and measurement occur due to model errors, e.g. in turbulence modelling, but also due to uncertainties in conditions of the experiment, e.g. uncertainties in geometry, boundary conditions or measurement error. Comparison between calculation and experiment should be performed after minimizing numerical errors according to best practice guidelines (Menter et al., 2002). So-called “best practice solutions” were included into the comparison, where numerical errors are reduced to maximum possible extend. Target values and comparison criteria have been evaluated in Hemström et al. (2005). This methodology is applied in the following for quantitative assessment of deviations between calculation and experiment.

For the quantitative comparison of the experimental and calculation results, three stationary experiments and one transient experiment at the ROCOM test facility were selected. The comparison was made between the calculated and measured data of the mixing scalar in the core inlet plane of the test facility. In this plane, 193 measurement positions are installed, one at the entry into each fuel assembly. Exactly at these positions, the calculated values for the mixing scalar were extracted from the calculations.

For the quantitative comparison different types of deviations were defined. These are

\[ \text{DEV1}_{i,t} = c_{c,i,t} - c_{m,i,t} \]  

where \(c_c\) is the calculated and \(c_m\) the measured value of the mixing scalar or the velocity at the position \(i\) and the time \(t\). DEV2 is an accumulated deviation at the certain position \(i\) over the important time span, i.e. when the perturbation is moving through the measurement plane. Two representations of the accumulated deviation were selected, the first is the absolute deviation.

\[ \text{DEV2}_{i,\text{ABS}} = \sum_{t=t_1}^{t=t_2} |\text{DEV1}_{i,t}| \]  

When considering the sign, also the direction of deviation reveals:

\[ \text{DEV2}_{i,\text{SIGN}} = \sum_{t=t_1}^{t=t_2} \text{DEV1}_{i,t} \]  

The averaging of the deviation DEV2 over all measurement positions leads to the average accumulated deviation DEV3_ABS and DEV3_SIGN. These values are absolute ones and are in same units as the mixing scalar (%). The relative deviations DEV3_rel are calculated relating the absolute deviation DEV3 to the integral perturbation introduced in the experiment into the core inlet plane.

5.1. Comparison for the stationary experiment ROCOM_stat01

Four CFD solutions obtained by different project partners were included into the comparison between measurement and calculation. Besides the three steady-state calculations, one transient calculation was included into the comparison (FZR_01tr). This calculation was performed with constant velocity boundary conditions at the inlet into the reactor pressure vessel. The calculated mixing scalar at the core inlet was averaged at the quasi-stationary concentration level in the same way as in the experiment. The results of the four calculations are shown in Fig. 4.

Table 4 shows the relative averaged deviation DEV3 for the different CFD solutions. The averaged perturbation should be close to 25% in the case of 100% perturbation in one of the four loops. However, the exact value of the perturbation depends on the velocity profile at the core inlet, which is unknown, but should be close to homogeneous in our case. The relative deviation considering the sign is very low for all calculations in comparison to the absolute deviation. That indicates, that a tilt between the measured and the calculated distributions leads to a significant compensation. The relative deviation based on the absolute values is in the order of 25% for all solutions.

Table 4

<table>
<thead>
<tr>
<th></th>
<th>Averaged perturbation (%)</th>
<th>DEV3_SIGN_{rel} Eq. C.4.7 (%)</th>
<th>DEV3_ABS_{rel} Eq. C.4.6 (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EXP</td>
<td>25.45</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>FZR</td>
<td>25.40</td>
<td>–0.345</td>
<td>26.1</td>
</tr>
<tr>
<td>AEKI</td>
<td>24.95</td>
<td>–2.046</td>
<td>28.5</td>
</tr>
<tr>
<td>VUJE</td>
<td>25.26</td>
<td>–0.906</td>
<td>25.1</td>
</tr>
<tr>
<td>FZRtr</td>
<td>24.53</td>
<td>–3.810</td>
<td>24.4</td>
</tr>
</tbody>
</table>
Table 5
Measured and calculated maximum values of the mixing scalar (including time of appearance)

<table>
<thead>
<tr>
<th></th>
<th>Total maximum (%)</th>
<th>Time of total maximum (s)</th>
<th>Maximum of the average (%)</th>
<th>Time of maximum of the average (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EXP</td>
<td>57.54</td>
<td>15.95</td>
<td>38.22</td>
<td>16.55</td>
</tr>
<tr>
<td>CFX5</td>
<td>60.34</td>
<td>14.50</td>
<td>33.12</td>
<td>17.00</td>
</tr>
<tr>
<td>CFX4</td>
<td>57.50</td>
<td>14.90</td>
<td>34.55</td>
<td>17.10</td>
</tr>
</tbody>
</table>

Quantitative comparisons are also performed for the experiments ROCOM_stat04 and ROCOM_stat09. FLUENT calculations have been performed for these experiments by VUJE and AEKI. The results are presented in Hemström et al. (2005). The tendencies are the same as observed for ROCOM_stat01.

5.2. Comparison for the slug mixing experiment ROCOM_02

Two solutions using the codes CFX-4 and CFX-5 were included into the comparison between measurement and calculation. Both calculations are obtained with the corresponding production mesh (tetrahedral mesh for CFX-5). In Table 5, the maximum values and the time point of maximum are compiled for the experiment and the two calculations. Both calculations predict the maximum of the mixing scalar slightly too early, 1 or 0.5 s, respectively. The value calculated by CFX-4 of the maximum is in good agreement with the measured one, while being overestimated in the CFX-5 calculation. Further, the accumulated normalized perturbation PERT was calculated according to:

\[ PERT = \frac{1}{(t_2 - t_1)} \sum_{i=t_1}^{t_2} \left( \sum_{i=1}^{n} \Theta_i(t) \right) \]

The integral perturbation is normalized in this way, that a value of PERT0 = 1.0 would be reached, when the mixing scalar over the whole considered time would be unity at all measurement positions, in other words we would have a full deboration over the full considered time. The integral perturbation was 0.146 in the experiment, 0.1566 in the CFX-5 calculation and 0.1124 in the CFX-4 calculation. In the CFX-4 calculation the introduced perturbation was underestimated in comparison to the experiment, whereas it was slightly overestimated in the CFX-5 calculation.

5.3. Conclusions on quantitative comparison between measurement and CFD-calculations

A detailed quantitative comparison of the results of CFD-calculations with measurements in the complex geometry of the model of a pressure vessel was performed. Different steady-state and transient experiments and calculations were considered. The comparison was mainly concentrated on the core inlet plane. Additionally, the velocity distribution in the downcomer was compared.

In the steady-state experiments, the different codes show the same global tendencies. The quantitative analysis revealed, that the spreading of the tracer in radial direction is underestimated in all presented calculations. The transient calculation with constant velocity in time shows the best agreement in the shape of the distribution of the mixing scalar in the core inlet plane and in the calculated maximum value. Further, the comparison of measurement positions, for which the coolant should flow through the sieve drum or around the sieve drum revealed great differences.

The pump start-up experiment ROCOM_02 was calculated by the codes CFX-4 und CFX-5. The time behaviour of the maximum and the average perturbation calculated by both codes is inside the confidence interval of 2σ during the main part of the considered time interval.

6. CFD calculations for the buoyancy-driven mixing experiments

CFD simulations have also been performed for the buoyancy driven mixing tests. Three of the Fortum PTS tests and one ROCOM generic buoyancy driven mixing case have been calculated.

CFX-5 calculations of the buoyancy mixing test D10M05 were performed by M. Scheurer and Höhne et al. (2006). The experiment was performed with injection of water with 10% increased density into the cold leg with 5% of nominal flow rate. The calculational grid includes detailed models of the sieve barrel, the core support plate and the rods modelling the core in the ROCOM test facility. The cold leg with the ECC-injection nozzle was added in order to capture the flow stratification in the cold leg which also influences mixing in the downcomer. The injection of water with higher density was performed from 5 to 15 s. Calculations were made using second order discretisation schemes in space and time. Two constant time steps (0.1 and 0.05 s) were used in combination with the SST turbulence model and scalable wall functions. Fig. 13 (up) shows the flow streamlines in the cold leg and the downcomer during and after the injection of the glucose solution. Fig. 13 (bottom) shows the distribution of glucose solution, detected by the tracer concentration, at the same time points. It can be seen, that during the injection, when the heavier water did not yet arrive in the downcomer, there is a more or less homogeneous distribution of velocity and tracer concentration around the RPV inlet nozzle. After the fluid with higher density has moved to the downcomer, the fluid motion is accelerated downwards, and a jet of heavier water is formed in the downcomer. The glucose solution flows downwards directly below the inlet pipe. These phenomena were also observed in the experiment.

Investigations on the influence of the turbulence model were performed by using a Reynolds Stress model (RST), based on
the omega equation. The RST model, although calculated with the larger time step (due to higher computer resources requirements), shows better agreement with measurement data.

However, quantitative comparison between measurement and calculation is difficult especially for the buoyancy driven mixing cases because of the fluctuating nature of the flow field and mixing pattern. It might be more useful to compare some averaged or integral parameters. This was done, e.g. for the Fortum PTS experiments.

The buoyancy driven turbulent flow with stratification and mixing in selected Fortum mixing tests was modelled by T. Toppila with the commercial CFD code FLUENT. The computation grid used for Fortum mixing test simulations is shown in Fig. 14.

The comparison was mainly based on the temperature/concentration data from the thermocouple locations near the pressure vessel wall and in the main (middle one) cold leg. The stratification and mixing in the main loop was studied using backflow ratio \( Q^* = Q_h/Q_{HPI} \) where \( Q_{HPI} \) is the cold-water injection flow rate and \( Q_h \) is the flow rate from back the downcomer to the main loop. The mixing of the cold water plume in the downcomer was studied using time-dependent maximum and average injection water concentrations at RPV wall at different vertical levels.

Fig. 15 shows a comparison of the measured and calculated the backflow ratio (up) and the maximum concentration of the cold HPI water in the downcomer (down) for the experiment #10 from Table 3. The main features of flow and mixing were quite well simulated; stratification in the main cold leg and the downcomer, backflow from the downcomer and side loops to the main loops and the flow field in the downcomer. However, the mixing of the cold water plume was not as effective in simulations as in real experiments. More grid size tests with transient simulations are needed to test the grid-independency of simulations.

Preliminary turbulence models tests with Realizable k-\( \varepsilon \) (Fortum) and RNG k-\( \varepsilon \) models (NRI) did not bring out any significant differences between models.

7. CFD simulation mixing tests at Paks NPP (VVER-440 reactor)

Additional test data were made available by Paks NPP and AEKI for the FLOMIX-R project (Elter, 2002; Rohde et al., 2005). The data have been gained from commissioning tests of Paks NPP performed in years 1987–1989. The tests addressed mixing among coolant loop flows in the downcomer and up to the core inlet in forced flow conditions. The goal of the tests
Fig. 14. Computation grid used for Fortum mixing test simulations.

Fig. 15. Backflow ratio (up) and cold HPI water concentration at pressure vessel wall at level $z = -1460$ (down) during simulations of experiment #10.

Fig. 16. VUJE model of VVER 440 V-213 reactor (left) and reactor internals (right).
was investigation of potential loop temperature asymmetry that might occur and significantly affect power distribution in the core. Paks mixing experiments were calculated by VUJE (FLUENT), AEKI (FLUENT) and TU Budapest (CFX5). It must be pointed out, that the calculations were performed for real plant geometry. Because of the 1:5 scaling of the test facilities, the computational cells are five times bigger, if the same number of cells is used in the real plant calculations as in the calculations for the test facilities. Moreover, the Reynolds numbers are much higher under the real plant conditions. This is an additional challenge for the real plant calculations. More detailed descriptions of the CFD analyses are given in Hemström et al. (2005).

Comprehensive computer models of VVER 440 reactor vessel were developed. They include all six loops with inlet nozzles and three baffles, whose purpose is to deflect the coolant injected into the reactor vessel from the safety injection tanks. Eight support consoles for the core barrel alignment were modelled as well. The production mesh used in the calculations of VUJE is shown in Fig. 16.

The sensitivity tests of the meshing showed that the calculation meshes did not meet the grid-independency criteria, inspite of the large node number. It was concluded that hybrid tetrahedral–hexahedral meshing which is available in the CFX-10 code seems to be more suitable for the pres-

Fig. 17. Comparison of calculated dimensionless core inlet temperatures with measured data for loop one (up) and deviations from measurement (bottom).
Table 6
Basic assumptions used in the calculations of Paks mixing tests

<table>
<thead>
<tr>
<th></th>
<th>AEKI</th>
<th>TU Budapest</th>
<th>VUJE</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD code used</td>
<td>FLUENT 6.1.22</td>
<td>CFX 5.5.1</td>
<td>FLUENT 6.1.18</td>
</tr>
<tr>
<td>Mesh</td>
<td>1,172,618</td>
<td>1,838,991</td>
<td>1,609,231</td>
</tr>
<tr>
<td>Elliptical perforated plate</td>
<td>Porous medium</td>
<td>Porous medium</td>
<td>Structure with 227 holes</td>
</tr>
<tr>
<td>Guide tubes</td>
<td>No</td>
<td>No</td>
<td>Second order</td>
</tr>
<tr>
<td>Discretisation scheme</td>
<td>First order</td>
<td>Second order</td>
<td>Second order</td>
</tr>
<tr>
<td>Wall function</td>
<td>Non-equilibrium</td>
<td>Scalable</td>
<td>Standard</td>
</tr>
</tbody>
</table>

Sure vessel geometry because of the lower number of mesh elements.

Comparison of the calculation results with the measurements was done only for those simulations which gave the best agreement (“production calculations”). Summary of basic assumptions used in the production calculations of Paks mixing tests performed by AEKI, TU Budapest and VUJE is presented in Table 6.

All calculations were performed as steady state. Results are shown in Fig. 17 in the form of temperature fields expressed in terms of dimensionless mixing scalars at the core inlet and compared with experimental data. For assessment of deviations between calculated results and measurements, maps of errors are presented as well.

The performed calculations showed that the coolant flows in sectors downwards in the downcomer, and the sectors remain also at the core inlet. The maximum of the mixing scalar is not located directly below the inlet nozzle, but an azimuthal shift of the maximum can be observed because of the asymmetric location of the inlet nozzles and because of the hydro-accumulator baffles. The effect of the baffles can be observed even at the core inlet.

The results show qualitatively good accordance with the measured data. However, the mixing is underestimated in the calculations (the calculated maximums of the mixing scalars are larger than the measured ones). Following conclusions on the CFD modelling of the flow distribution in the RPV of a VVER-440 type reactor were drawn:

- Good numerical convergence was achieved with basic solver and solver settings based on residuals of continuity and momentum equations (using first order upwind discretisation scheme and k-ε or RSM turbulence model).
- Some numerical fluctuations of the flow field were observed below the ECC baffles in the downcomer, possibly due to some non-stationarity of the flow field.
- Modelling of detailed internal geometry (e.g. flow baffles) may have a noticeable influence on results.
- ECC water baffles guide the main flow and generate turbulence effects in the downcomer.
- The alignment drifts have only local effect.
- Accurate model of the perforated elliptical bottom (modelling the elliptical perforated bottom as solid structure rather than using a porous medium) provides more realistic flow pattern and improves the accuracy of the calculation.
- Using pressure outlet boundary condition gives better numerical convergence than outflow boundary.
- Boundary layer at the walls of downcomer should be less than 3 mm, to keep y⁺ parameter less than 1000.
- High order methods decrease the numerical stability of the solution.

8. Conclusions

8.1. Conclusions from sensitivity tests according to the BPG

The most important conclusions from the sensitivity tests made in the CFD simulations of steady-state mixing and flow distribution, transient slug mixing and buoyancy driven mixing within the EC project FLOMIX-R are presented below. Sensitivity tests are an important part of applying the best practice guidelines for CFD in terms of quality assurance. However, as the test facilities for which the validations were made are quite different, it is in some cases difficult to draw general conclusions. The conclusions are also based on calculations that have not been shown to be grid independent.

8.1.1. Grid size

The aim is to make CFD calculations that give grid-independent solutions, i.e. results that do not change when the grid is refined further. A grid-independent solution can be defined as a solution that has a solution error that is within a range that can be accepted by the end-user, in view of the purpose of the calculations. According to the BPG, calculations must be made with at least three different grids in order to be able to quantify the grid-dependence of the calculations. The two finer grids also have to be made using a complete refinement of the nearest coarser grid, to get an objective measure of the grid-dependency. For example, if a calculation is made with 200,000 cells, calculations also have to be made with 1,600,000 cells and 12,800,000 cells. Due to limited computer resources, this procedure has not been possible to follow for any of the slug mixing test cases in FLOMIX-R. A transient calculation for a coarse grid of around 200,000 cells takes in the order of 1 week to perform. The solution errors have therefore not been quantified. Developments in computer technology will make these quantifications of solution errors possible within a few years. The number of cells required for grid-independency can only be guessed. Several million cells are probably needed for most of the transient test cases in
FLOMIX-R. Inspite of fully grid-independent solutions, solutions on so-called production meshes were used to assess the agreement between measurement and calculations. The production meshes were an optimum in mesh refinement what could be reached.

8.1.5. Turbulence models

It is well known that the choice of inlet boundary position and inlet boundary condition (turbulence level, variation in inlet velocity field in space and time) can have a big influence on the flow pattern far downstream from the inlet. Sensitivity tests have been made in some cases, otherwise the inlet was put far upstream from the downcomer at a position where the flow conditions were well known.

For the ROCOM steady-state mixing case, results were not sensitive to different turbulence intensities at an inlet positioned close to the downcomer. Slightly different results were achieved if the tracer concentration profile was changed at the same inlet position. Modelling of the four cold legs, bends included, gave only slightly different results compared to having an inlet close to the downcomer.

8.1.8. Height of wall-adjacent cells

One sensitivity test was made for the height of the wall-adjacent cells for the Vattenfall steady-state case. The maximum $y^+$ values were very high with the grids used, up to around 5000. The grid was therefore refined at the wall-adjacent cells. However, calculations with these lower and more optimum $y^+$ at the walls gave worse results for the Vattenfall steady-state case. This might indicate that the non-equilibrium wall function used does not work properly for the type of boundary layers present in the downcomer, especially those close to the flow-impingement where the jet from the inlet pipe hits the downcomer wall. For the ROCOM production mesh the $y^+$ value in the downcomer was 65, and therefore below the recommended value of maximum 100.

8.1.9. Code

The commercial codes CFX4, CFX5 and Fluent 6 were used. For both the Vattenfall slug mixing case and the ROCOM_02 slug mixing case the differences between the results from Fluent 6 and CFX5 were significant, inspite of the fact that the calculations were run with exactly the same grids. For the ROCOM steady-state mixing case, the differences were smaller. However, as none of the calculations are probably grid-independent, one cannot expect to obtain the same results from the codes, as the numerics in the codes are different. Another difference between
the two codes is that exactly the same wall functions could not be applied.

8.1.10. Numerical schemes

At least second order schemes should be used, both in space and time. For some applications there were convergence problems (unstable solutions) when using second order schemes. This might indicate that there are also low-frequency fluctuations in the measurements, which cannot be resolved by a steady-state calculation. For these cases a time-averaged flow field might have to be calculated with a transient calculation.

8.2. Comparisons with measurements

8.2.1. Steady-state mixing and flow distribution

In general, the flow pattern, velocity distribution in the downcomer and mixing scalar distributions at the core inlet are well predicted in the CFD calculations. The velocity field in the downcomer has inhomogeneous character with maximum downwards flow components in the regions in-between the inlet nozzles. A clear sector formation of the flow in the downcomer is seen. This leads to maximum mixing scalar values at the core inlet of 92–99%. That means a part of the fluid remains almost unmixed. The re-distribution of the velocity field and mixing scalar distribution in the case of asymmetric flow conditions is also qualitatively well reproduced in the calculations.

Finer grids in the CFD simulation tend to give better results. Also modelling of perforated sheets (such as the drum in the downcomer) as real structure rather than porous medium improves quality of results. Influence of porous medium as a substitute of a perforated sheet can be, in some extent, controlled by proper definition of direction-dependent resistance of the porous medium. Other investigated effects (turbulence model, wall function, position of outlet boundary) do not have an unambiguous influence on results. In general, the mixing along the flow path in the downcomer is under-predicted in all calculations. Disturbance in the inlet boundary conditions has significant impact on the flow pattern.

8.2.2. Slug mixing transients

For the ROCOM slug mixing transient the qualitative agreement with measurements is good. The position of the lowest boron concentrations was captured fairly well. Quantitative good agreement with the level of the measured lowest boron concentrations was achieved. Considering the agreement of the measured and calculated boron concentration values at local positions, the deviations are larger than for the global minimum. For the ROCOM buoyant slug mixing transient, the local concentration was over-predicted, i.e. mixing was under-predicted.

For the Vattenfall slug mixing transient, the minimum average boron concentration at the core inlet was captured very well by the CFD calculations. Also, the minimum boron concentration at the core inlet was captured very well. The distribution of low boron concentration across the core inlet plane was not, however, accurately modelled by the CFD calculation. There is mainly a rotational displacement of the concentration field. This could be relevant in reality, as a core is primarily different radially, as far as enrichment and reactivity are concerned.

In the Gidropress slug mixing transient the calculated minimum of the average core inlet temperature was lower than the experimental one. This was probably a consequence of the fact that the heat exchange between the cold slug and the warm walls was neglected in the CFD calculation. Other possible reasons are the pre-mixing in the main coolant pump simulator, which is difficult to model, and the not exactly known initial position of the slug boundary.

The CFD calculations for the FORTUM PTS buoyant transient mixing case modelled the main features of the flow quite well. Quantitatively there was some poor agreement for mixing in the main loop and for the mixing of the cold plume in the downcomer.

The conclusions from the various calculations made were not unanimous. The results are promising, however, better agreement with measurements is needed for a CFD calculation to be an equivalent competitor to model tests. The continuous developments in computer capacity and in software capabilities will allow more extensive calculations to be made, enabling a reduction in errors from all sources, and increase the accuracy of CFD calculations. A methodology of quantitative comparison between measurement and calculation was developed and applied. It provided very useful information concerning a quantified engineering error assessment which can be used, e.g. in reactor physics calculations concerning the consequences of boron dilution transients as a safety surcharge.

8.3. Development needs

This section gives a summary of the conclusions drawn from the calculations concerning what is needed in the future to obtain more accurate CFD calculations for boron dilution transients and mixing in pressurized water reactors. As shown earlier in this report agreement with measurements is not satisfactory in all cases. In order to get results from CFD calculations that are in better agreement with measurements, the following points have to be considered:

- More computational cells. No fully grid-independent solutions were obtained due to constraints in computer resources. More detailed modelling of internal structures might also be needed, which leads to additional cells.
- Shorter time steps might in some cases be needed.
- More advanced turbulence models, especially models that can cope with the specific features of the flow fields present in these applications, such as accelerating flow, flow-impingements and buoyancy. Further-improved Reynolds Stress Models or LES models (or hybrids between the two) might be needed to get better agreement with measurements. Better models of wall boundary layers (i.e. wall functions) are also probably needed.
Some steady-state cases have to be run as transients, as low-frequency fluctuations cannot be captured by a steady-state calculation. A transient calculation takes at least an order of magnitude longer time to perform than the corresponding steady-state calculation.

The computation time needed for a grid- and time step-independent CFD calculation using an advanced turbulence model is today too long (in the order of months). To be able to perform such calculations in a shorter period of time, the following aspects are important:

- Faster and more accurate solvers and numerical schemes.
- Automatic time stepping. The CFD code should choose the optimal time step (within specified limits) and relaxation for minimizing computation time and ensuring convergence to specified residual levels.
- Improvements of the grid generation process in order to be able to, in an easier way than today, make high-quality hexahedral grids that would produce more accurate solutions with fewer cells.
- Grid adaptation (refine and coarsen) during transient runs to get refinements where there are strong gradients of (especially) concentration will also produce more accurate solutions with fewer cells.
- Refined porosity models can reduce the need for cells.

The best practice guidelines for CFD have increased the awareness for what is needed to produce high-quality CFD calculations. These guidelines should be extended with more detailed recommendations, for example, which turbulence model to use for different types of flow situations. The CFD code should also help the user to apply the best practice guidelines, for example, concerning quality checks for grid and solution.

Inspite of the large amount of work performed in FLOMIX-R there are still questions unanswered about the capability of CFD codes to model boron dilution transients and mixing in primary system in pressurized water reactors. The continuous development in computer capacity and in software capabilities will continuously increase the ability to make accurate CFD calculations.

In general, it can be concluded, that CFD codes can be applied for calculations of turbulent mixing in one-phase flow in an engineering way.

References


