CFX-10 and RELAP5-3D Simulations of Coolant Mixing Phenomena in RPV of VVER-1000 Reactors

Fulvio Terzuoli, Fabio Moretti, Daniele Melideo, Francesco D’Auria
University of Pisa – Department of Mechanics, Nuclear and Production Engineering
2, Via Diotisalvi, 56126, Pisa, Italy
f.terzuoli@ing.unipi.it, f.moretti@ing.unipi.it, daniele.melideo@ing.unipi.it, f.dauria@ing.unipi.it

Oleksandr Shkarupa
International Nuclear Safety Center of the Taras Shevchenko National University of Kyiv
60, St. Volodymyrs'ka, room 200, 01033, Kyiv, Ukraine
shkarupa@i.kiev.ua

ABSTRACT

The present paper deals with numerical analyses of coolant mixing in the reactor pressure vessel of a VVER-1000 reactor, performed with the ANSYS CFX-10 CFD code and with the RELAP5-3D system code. In particular, the attention focused on the “swirl” effect that has been observed to take place in the downcomer of such kind of reactor, with the aim of assessing the capability of the codes to predict that effect, and to understand the reasons for its occurrence. The results have been compared against experimental data from V1000CT-2 Benchmark.

1 INTRODUCTION

Coolant mixing phenomena occurring in the pressure vessel of a nuclear reactor constitute one of the main objectives of investigation by researchers in the nuclear reactor safety field. In-vessel flow mixing has been demonstrated to play an important role in reactivity-induced accidents in Pressurized Water Reactors (PWR) initiated by coolant deboration or boron dilution events, since they reduce the consequences of reactivity insertion. Similarly, also thermal mixing phenomena may affect the reactor physics response during asymmetric operation events (such as Main Steam Line Break accidents) and reduce the possibilities of thermal shocks. Predictive analysis of mixing phenomena is strongly improved by the availability of computational tools able to simulate the inherent three-dimensionality of such problem, such as system codes with three-dimensional capabilities, and CFD codes.

Several research activities have been carried out to investigate the mixing phenomena occurring inside the Reactor Pressure Vessel (RPV) of PWRs, and to assess the computational tools for the simulation of flow mixing. Research projects, like EUBORA (Concerted Action on Boron Dilution Experiments [1]), ECORA (Evaluation of Computational Fluid Dynamics Methods for Reactor Safety Analysis [2]) and FLOMIX (Fluid Mixing and Flow Distribution in the Primary Circuit [3]), were conducted within the European Community (EC) Framework Programmes, with the purpose of developing and assessing computational tools, especially CFD codes, which are assumed to be the most suitable tool for the predictive analyses of flow mixing problems.
The International Standard Problem No. 43 [4], organized by the OECD and based on the experimental data collected at the University of Maryland – College Park (UMCP) mixing facility, has constituted an important benchmarking for CFD code users and developers. Furthermore, the second V1000CT Benchmark, “Coolant Mixing Tests and Main Steam-Line Break (MSLB)” [5], supported by OECD and USNRC, based on VVER-1000 reactor and currently in progress, also constitutes a chance for assessing CFD code capabilities in predicting in-vessel mixing.

The present paper deals with the CFD code assessment activity currently ongoing at University of Pisa, in particular focuses on the application of the ANSYS CFX-10 code to the simulation of mixing phenomena occurring inside a VVER-1000 RPV. The application of the RELAP5-3D system code to the same problems, is presented as well.

The CFD simulations were run on computational grids representing a VVER-1000/320 reactor, and characterized by rather high level of geometrical detail, although not fine enough to yield mesh-independent solutions. The simulations addressed a thermal mixing problem referred to the final steady-state conditions of the Steam Generator (SG) isolation scenario of the V1000CT-2 Benchmark – 1st exercise. Temperature distribution and mass flows were calculated and compared against results form RELAP5-3D [6] and experimental data. All CFD calculations were run in parallel mode on a small Linux-cluster.

2 CFD NUMERICAL MODEL

2.1 Computational Domain

The VVER-1000 is a 4 loop PWR, whose core contains 163 hexagonal Fuel Assemblies (FA) with hexagonal layout. The coolant flows into the core through the perforated barrel bottom and FA support columns serving as flow distributors. The FA support columns are perforated and inserted into corresponding holes of the core support plate and welded at the top so that no flow passes outside the columns. Thus, the coolant flows through the slots into the columns, and then into the FA. Some geometrical differences exist between VVER-1000/320 reactor belonging to different Nuclear Power Plants (NPP). In this work reference was made to the Kozloduy NPP Unit 6 described in the Benchmark.

The computational domain chosen for the CFD simulations includes the inlet nozzles region, the Downcomer (DC) and the Lower Plenum (LP) up to the core inlet plate, as shown in Figure 1-a. A sketch of the overall three-dimensional CAD model is shown in Figure 1-b, this model includes many complex geometrical details, such as the eight consoles in the DC between the internal RPV wall and the barrel, and the solid support columns between the elliptical shell and the core inlet plate (Figure 2-a). The consoles are equally spaced along the azimuthal direction, and their layout is symmetric with respect to the core symmetry planes. Smaller details, like the 1344 holes on the elliptical shell (blue region in Figure 2-b) and the holes on the perforated columns (violet region in Figure 2-b), were not represented and were accounted for by additional pressure losses in the numerical model. The CAD model includes an outlet volume above the core inlet that simulates the core region and allows to apply a pressure-controlled outlet boundary condition.
2.2 Grids Generation

The computational grids were created with the ANSYS ICEM CFD package. As a first meshing strategy, the use of tetrahedral elements all over the computational domain was chosen. This choice is usually the most versatile when dealing with complex geometries. The use of hybrid meshes (hexahedral cells in the DC, and tetrahedral cells in the LP) has also been considered: it allows to reduce the total number of cells, but usually implies more difficulties. The preparation of hybrid grids is currently in progress at University of Pisa.

Two different grids have been used, the former counts around 4.2 million tetrahedral elements (three millions only in the LP) and contains about 0.9 million nodes, while the latter is an enhanced version of the former: 3 prisms layers (1mm first layer thickness, 1.2 expansion ratio) have been added to better predict the flow in the areas adjacent to the solid boundaries of the inlet nozzles, the barrel, the consoles and the vessel. The resulting grid, constituted by ~0.7 million prismatic and ~4.2 million tetrahedral elements, contains about 1.3 millions nodes. Some particulars of the tetrahedral mesh are shown in Figure 3. The smallest tetrahedral cell size is lower than 30 mm. Despite these large numbers the prepared meshes are relatively coarse, representing a compromise between accuracy and computing time in relation to the available computing power. It has to be remarked that, as suggested by the Best Practice Guidelines [8], a mesh should be proved to yield grid-independent results, and systematic mesh sensitivity analyses should be performed. However, this task is almost always impracticable when dealing with such complex geometries.
2.3 Problem Description

The performed calculations are referred to Exercise 1 of the V1000CT Benchmark – Phase 2, which represents a SG isolation transient reproduced at the Kozloduy-6 Nuclear Power Plant. Reference is made to the final steady-state conditions of the scenario addressed in the Benchmark. Unsteady simulations of the whole transient were not performed since the available computing power (a Linux-cluster of 16 processors, 2.8 MHz CPU and 2 MB RAM each), was not sufficient to obtain acceptable results in reasonable time.

The problem is characterized by asymmetrical temperature distribution in the cold legs. In particular, owing to the SG isolation, the loop #1 inlet temperature is about 12-13 K higher than in the other loops, therefore one sector of the RPV is affected by a temperature perturbation. The simulations were aimed at predicting the thermal mixing between flows coming from the 4 loops, and the temperature spatial distribution at core inlet. Figure 4 shows the reactor core layout, along with the FA equipped with thermocouples (at channel outlet). It also shows that the vertical symmetry planes of the cold legs do not coincide with the core symmetry planes, rather they are rotated around the vertical axes by 7° counter-clockwise.

Figure 4: Reactor layout - a) core inlet map showing instrumented channels (in yellow); b) mismatch between core symmetry planes and cold legs symmetry planes.

Proceedings of the International Conference Nuclear Energy for New Europe, 2006
2.4 Simulations Setup

The ANSYS CFX 10.0 simulations were performed solving for the continuity, momentum conservation and heat conservation equations, in their incompressible and steady-state formulation. The turbulence was accounted for with the Reynolds-averaged Navier–Stokes (RANS) approach. Namely the two-equation $k-\varepsilon$ turbulent model was used, combined with Scalable Wall Functions.

Water physical properties are those corresponding to the temperature and pressure conditions (543 K and 16 MPa) specified by the Benchmark. Upwind and High Resolution discretization schemes have been used for the advection terms in the transport equations. The buoyancy effects have been neglected, owing to the small temperature variations and to the high Reynolds numbers.

As boundary conditions, uniform average velocities and static temperatures have been imposed at the inlet boundaries, according to the Benchmark data, as shown in Table 1. The inlet turbulent parameters are: 5% turbulent intensity, 10 viscosity ratio.

Table 1: Inlet boundary conditions.

<table>
<thead>
<tr>
<th>Loop #</th>
<th>Average inlet velocity [m/s]</th>
<th>Inlet static temperature [K]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10.57</td>
<td>555.35</td>
</tr>
<tr>
<td>2</td>
<td>10.54</td>
<td>543.05</td>
</tr>
<tr>
<td>3</td>
<td>10.51</td>
<td>542.15</td>
</tr>
<tr>
<td>4</td>
<td>10.85</td>
<td>542.35</td>
</tr>
</tbody>
</table>

According with the domain modelling, additional pressure losses have been assigned to the perforated columns (outer side region), and to the perforated elliptical shell region, in yellow and green respectively in Figure 5.

Figure 5: Model sub-regions - Additional pressure losses have been assigned to the perforated columns (outer side region), and to the perforated elliptical shell region.

The additional pressure losses have been imposed by defining source terms in the momentum conservation equations, according to Eq. (1):

$$ S_i = -\frac{1}{2} k \rho |U| |U_i| $$

where $k$ is the resistance loss coefficient, and $i$ indicates the three Cartesian directions. Table 2 shows the values assigned to $k$, so as to obtain an overall pressure drop in agreement with the experimental data. To evaluate the influence of the pressure loss coefficient, a

Proceedings of the International Conference Nuclear Energy for New Europe, 2006
simulation has been performed with a ten times higher value for the perforated columns region coefficient.

<table>
<thead>
<tr>
<th>Table 2: Used resistance coefficients.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Perforated elliptical shell region (directional model)</strong></td>
</tr>
<tr>
<td>$k$ for streamwise direction (vertical axes)</td>
</tr>
<tr>
<td>$\text{multiplier for transverse directions}$</td>
</tr>
<tr>
<td><strong>Perforated columns region (isotropic model)</strong></td>
</tr>
<tr>
<td>$k$ for any direction</td>
</tr>
</tbody>
</table>

3 RELAP5-3D NODALIZATION

A RELAP5-3D nodalization (see Figure 6), developed at the University of Kiev [6], was used for simulating the problem described above, with purpose of assessing the system code capabilities to simulate the mixing phenomena through a code-to-code comparison.

![Figure 6: University of Kiev RELAP5-3D nodalization of the Kozloduy-6 reactor [6].](image)

The nodalization allows the coupling between thermal-hydraulic model and neutron kinetic model, however such capability was not used for the purposes of the present study.

Each of the 163 fuel assemblies was individually modelled with a thermal-hydraulic channel subdivided into ten layers.

4 RESULTS

Some obtained results are shown in Figure 7, and compared against plant data. The best agreement with plant data is obtained for CFX simulations with Upwind scheme (Figure 7-b), both in terms of perturbation “shape” and of temperature spatial gradients. RELAP5-3D results (Figure 7-c) appear quite similar to CFX – Upwind results. CFX – High Resolution results (Figure 7-a) show much steeper temperature gradients, thus indicating an underestimation of the turbulent mixing. This may suggest that the overall CFD model tends to underestimate mixing, and that the use of low-order discretization scheme like the Upwind, which are affected by greater numerical diffusion, may have a compensating effect on such deficiency, thus improving the results.
However, as can be seen in Figure 7, the correct counter-clockwise rotation (the so-called “swirl effect”) of the temperature perturbation is not correctly predicted by the simulations, since they provide no rotation of the coolant. The swirl effect has been observed at Kozloduy-6 [5] and in several other VVER-1000 type plants. It consists in a rotational component of the whole flow (which is, for instance, 15-20° counter-clockwise for Kozloduy NPP), which in turn strongly affects the temperature (or any other transported scalar) distribution at core inlet.

Figure 7: Temperature distribution at FA inlets – a) CFX-High Resolution with tetrahedral grid; b) CFX-Upwind with tetrahedral grid; c) RELAP5-3D; d) Kozloduy-6 data.

The relevance of the asymmetric cold leg orientation with respect to the core (see Figure 4) to the addressed phenomenon can be excluded since it is known that the swirl effect has been observed also in VVER plants having symmetric configurations. This effect has not yet been fully explained, although some relevant suggestion came from recently published works, like that of Bieder et al. [9]. The authors concluded that the swirl effect is due both to some hydraulic instabilities and to the small discrepancies existing between the design layout of the cold legs and the real plant layout (angular locations differ of a few percents). Others possible contributions to the addressed phenomenon could be the presence of secondary motions in inlet flows (Moretti et al. [7]) and the presence of the two ECC nozzles, since they affect the local flow resistance. Investigations on this possible influence are currently ongoing at University of Pisa.

It has to be remarked that the result obtained using CFX with the High Resolution scheme and the grid with prisms layers, provide a counter-clockwise rotation of the coolant as shown in Figure 8-a, however, the use of high order schemes lead sometimes to numerical
instabilities and unphysical behaviour, as should be suggested by Figure 8-b. To better understand this phenomena, further investigations need to be undertaken.

Figure 8: Temperature distributions calculated with CFX using High Resolution scheme and grid with prisms layers - a) FA inlets; b) Vessel walls

The comparisons between the average coolant temperatures at the FA inlets calculated with CFX (for both Upwind and High Resolution schemes, both grids and both values of resistance loss coefficient) and RELAP5-3D are shown in Figure 9 and Figure 10; they are compared against plant data, however the absence of the “swirl” effect from all the results makes difficult the comparison of the local temperature values. These Figures also shows that all the performed calculations provide a systematic overestimation of the temperature values.

Figure 11 and Figure 12 show the comparisons between the mass flows at FA inlets, where they have been renumbered from the outer hexagonal row to the inner one in a spiral-like sequence starting from the FA #1, in a counter-clockwise direction.

Figure 9: Temperature values at FA inlets – CFX-Upwind.

Proceedings of the International Conference Nuclear Energy for New Europe, 2006
Figure 10: Temperature values at FA inlets – CFX-High Resolution and RELAP5-3D.

Figure 11: Mass flows values at FA inlets – CFX-Upwind.

Figure 12: Mass flows values at FA inlets – CFX-High Resolution.
As can be seen in Figure 9 and Figure 11, when using the Upwind scheme the presence of prisms layers has no appreciable effects on temperatures and mass flows distributions, since the calculated temperature values differ less than 0.1 K, while the mass flows differ less than 0.2 kg/s (which corresponds to 0.05%). This is not true when using the High Resolution scheme, in fact, although the mass flows values are not affected by the presence of prisms layers (see Figure 12), the average coolant temperature values differs considerably (see Figure 10), since in FA #66 and #32 the difference is higher than 10 K and the average temperature difference is about 1.6 K. However this result should be caused by numerical instabilities and unphysical behaviour as mentioned above.

As shown in Figure 9, the variation of the resistance loss coefficient has no appreciable effects on the temperature values at the FA inlets, since they differ from the calculation with nominal coefficient value (named “Tetrahedral grid + Prisms layers” in the Figures) by less than 0.2 K. This is not true for the mass flows (Figure 11), since the calculations provide values with a maximum difference of 17.6 kg/s (which corresponds to 13%) and an average absolute difference of 5.7 kg/s (4.7%). The model with increased resistance loss coefficient provides values about 10% higher for the FA from #1 to #36 (located in the outer hexagonal row), while it provides nearly the same values from #37 to #72 (located in the second hexagonal row), elsewhere the values are about 5% lower. In conclusion, it resulted that, using the Upwind scheme, the increased resistance loss coefficient has no appreciable effects on the temperature values, while it affects the mass flows, which are increased in the outer FA and reduced in the inner.

5 CONCLUSIONS

The CFX-10 and RELAP5-3D codes have been applied to the simulation of a steady-state thermal mixing problem on a computational model representing the RPV of a VVER-1000 reactor. The results were compared against the experimental data. The calculations were performed in order to evaluate the capabilities of the codes to predict in-vessel mixing problems. They included the use of prisms layers strategy for a better turbulence wall treatment, and sensitivity analyses on numerical discretization scheme and resistance loss coefficient.

Concerning the prediction of the main flow phenomena and of the temperature distribution, the CFD calculations yielded a qualitatively good agreement with the experimental data, however they turned out to underestimate the effectiveness of the turbulent mixing. This may be related to the inability of two-equation turbulent models to accurately simulate the complex flow patterns occurring inside a RPV. A compensating effect was observed when the lower order discretization scheme was used, since the higher numerical diffusion enhances the predicted turbulent diffusivity.

The results of the RELAP5-3D simulation are similar to those obtained from the CFX simulations with the lower order discretization scheme, both from a qualitative point of view and in terms of turbulent mixing.

Both computational tools resulted to be unable to predict the “swirl effect”. However, the physical phenomenon itself and its causes have not yet been fully understood, thus the attempt to simulate the effect is premature and should be postponed until a deeper physical comprehension is achieved.

The lower plenum modelling (simplification of the smallest geometric details, additional pressure losses, etc.), the role played by numerical diffusion and the turbulence modelling are the identified issues requiring further investigation and efforts.
REFERENCES


