Recommendation on Use of CFD Codes for Nuclear Reactor Safety Analysis

Author: Dominique Bestion
Contribution: Alain Martin
             Florian Menter
             Marc Boucker
             Sylvain Pigny
             Martina Scheuerer
             Matthias Heitsch
             Ulrich Rohde
             Sander Willemsen
             Henri Paillère
             Dave Sweet
             Michele Andreani

Dissemination level: PU
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>1</th>
<th>USE OF SINGLE-PHASE CFD</th>
<th>3</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Nuclear Reactor Safety Problems where CFD is recommended</td>
<td>4</td>
</tr>
<tr>
<td>1.2</td>
<td>How to adjust the ECORA BPG to large scale and reactor problems</td>
<td>7</td>
</tr>
<tr>
<td>1.2.1</td>
<td>PTS Validation 1: Jet impingement with heat transfer</td>
<td>7</td>
</tr>
<tr>
<td>1.2.2</td>
<td>PTS Demonstration 1: UPTF liquid-liquid mixing</td>
<td>8</td>
</tr>
<tr>
<td>1.2.3</td>
<td>Containment Demonstration: Analysis of two PANDA tests</td>
<td>8</td>
</tr>
<tr>
<td>1.2.4</td>
<td>Recommendations</td>
<td>11</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>2</th>
<th>USE OF TWO-PHASE FLOW CFD</th>
<th>12</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Nuclear Reactor Safety Problems where CFD is recommended</td>
<td>12</td>
</tr>
<tr>
<td>2.2</td>
<td>How to adjust the ECORA BPG to reactor relevant two-phase problems</td>
<td>14</td>
</tr>
<tr>
<td>2.2.1</td>
<td>Verification Test-Cases for Two-Phase Flow Codes</td>
<td>14</td>
</tr>
<tr>
<td>2.2.2</td>
<td>Control of errors in two-phase flows CFD simulations</td>
<td>15</td>
</tr>
<tr>
<td>2.2.3</td>
<td>Concluding remarks</td>
<td>17</td>
</tr>
</tbody>
</table>

| 3 | REFERENCES | 18 |
1 USE OF SINGLE-PHASE CFD

Reactor Safety Analysis related to both Pressurised Water Reactors (western type and VVER type) or Boiling Water Reactors mainly relied in a first step on system codes where the primary (and secondary) flows are modelled with a rather coarse nodalisation including about $10^3$ mesh points or “control volumes”. However some safety issues were clearly identified where a much finer resolution of the simulation tools was required. These issues are often related to situations where the 3D aspects of the flow and the geometrical effects have a significant influence on the safety criterion. Turbulent mixing is a common feature of these flows and the degree of mixing controls the result which directly affects the safety. Single phase CFD tools are then required which may model small scale mixing phenomena with a fine space resolution including $10^5$ to $10^7$ mesh points. The experimental investigation may also give the answer to the safety problem if the industrial geometry is strictly respected. Reliable simulation tools, after having been validated for each basic flow process and for some prototypic geometry, may allow much rapid answer to a new problem and/or a new geometry. Such single phase CFD tools exist and are commonly used in many industrial sectors and are now applied for Nuclear Reactor safety of for design purposes.

The principal interest of industrial computational fluid dynamics (acronym CFD) consists mainly in the capability to obtain at a lower cost, valuable information on some physical phenomena. Numerical simulations of industrial processes enable to test virtually any configuration, from both qualitative and quantitative points of view, and thus to evaluate and/or discriminate different designs according to pre-determined criteria. These criteria may be linked to economical or technological constraints, and/or to safety and environmental issues. Concerning safety issues in the nuclear industry, CFD has now been recognized as an important tool, as discussed in the IAEA/NEA workshop of November 2002.

In order to produce trustworthy studies on various problems, both “in house” and “commercial” software for thermal hydraulics and industrial fluid mechanics were developed and validated. Commercial codes such as CFX and FLUENT are widely and increasingly used in nuclear reactor safety applications. In the nuclear reactor community, for example, the SATURNE code is developed at EDF and the TRIO-U code and CAST3M code are developed at CEA for single phase flow whereas the NEPTUNE platform developed by CEA and EDF includes a two phase flow CFD tool. Two-phase models are also available in commercial codes. A multi-year validation study of the multi-phase capability in CFX-5 for reactor safety applications is currently carried out in a German research project coordinated by GRS (CFD Kompetenzverbund Reaktorsicherheit - German CFD Network in Nuclear Technology). In the 6th Framework Programme, the NURESIM proposal will be an integrated project aiming at developing a common nuclear reactor simulation platform, which will also include two-phase CFD modules. All these codes are engaged in a qualification process in the field of nuclear thermal hydraulics in order to ensure that the software is effectively able to produce relevant results in a clearly defined application field.

Developing and testing the tools has required intensive work on complex physical modelling and numerical schemes, both domains being closely linked in the CFD field. In parallel, experimental data bases are created, including Separate Effect Tests and some real size industry-like experiments; they are used to validate the physical modelling implemented in the codes. A comprehensive measurement data base on turbulent mixing inside the reactor pressure vessel has been created within the EC project FLOMIX-R on fluid mixing and flow distribution in the primary circuit of PWR. This data base gained from experiments at various test facilities representing different types of European reactors (German KONVOI, Westinghouse and Framatome –ANP PWR, VVER) is made available for CFD code validation purposes.

Although the design of the first PWRs was mainly based on an experimental approach in particular for evaluating the loads applied on the structures, present numerical tools are now able to model the structures even with complex geometry using a 3D numerical model and to solve the complex physical aspects of the flows. Different flow features take place in normal operating conditions, like jet impact, flow reversal, piping swirl effect, and in accidental conditions, buoy-
ancy effects or dilution problems are encountered, either in the primary system or in the reactor containment in the event of a Loss of Coolant Accident. Now, for the new reactors such as the EPR, the experimental approach is coupled to the numerical approach to provide elements required by the design.

Although not addressed in the framework of the ECORA project, other thermal-hydraulic phenomena require the use of CFD for safety assessment: for example, hydrogen combustion issues (addressed in the 4th FP projects HDC (Bielert et al., 2001) and HYCOM (Bielert et al., 2003) or the 5th FP EXPRO http://batchelor.uc3m.es/expro/expro.html), as well as phenomena representative of Generation IV reactors such as Gas-Cooled Reactors (Decay heat removal phenomena, depressurisations, thermal fatigue, etc) – see for example IAEA-TECDOC-1163, IAEA-TECDOC-1382 or also the proceedings of the HTR-2002 conference for examples of application of CFD to such problems. The development of Best Practice Guidelines for those particular applications is also needed, and would prove beneficial to the overall quality of the simulations.

1.1 Nuclear Reactor Safety Problems where CFD is recommended

Application of single phase CFD for NRS problems requires that the simulation tools have reached a sufficient degree of maturity and reliability and that the users have also a high expertise in selecting appropriately the turbulence modelling, the boundary conditions, the meshing, the numerical scheme options and in drawing clear conclusions on safety issues. Validity domains or reliability domains of the codes have to be established with respect to a certain number of criteria. In order to increase the confidence in numerical solutions, the codes are qualified on a wide set of validation test cases, ranging from analytical to more industrial cases. Such a maturity of both the tool and the user is reached or can be reached in a reasonable term and single phase CFD can be a very powerful tool for better understanding physical behaviour and one may recommend using it for a number of flow configurations encountered in safety analyses:

- Boron dilution
- Mixing of cold and hot water in Steam Line Break event
- Hot-leg temperature heterogeneity
- PTS (pressurised thermal shock)
- Counter-current flow of hot steam in hot leg for severe accident investigations of a possible “Induced Break”
- Thermal fatigue (e.g. T-junction)
- Hydrogen distribution and combustion in containment

In addition to these problems related to the present generation of water reactors, there is also a number of issues for advanced (including Gas-Cooled) reactors. One may give a few examples:

- Natural circulation in LMFBRs
- Coolability & Flow induced vibration of APWR radial reflector
- Flow in lower plenum of ABWR
- Depressurisation of a GCR
- Decay heat removal in a GCR
- Thermal loading on structures, etc.
- Containment integrity (peak pressure) during the long-term cooling of innovative reactors with passive safety systems, particularly in a BDBA scenarios (5th FWP project TEMPEST, Wichers and Huggenberger, 2004).

Boron mixing

A potential risk of a reactivity excursion accident exists in association with boron dilution events or overcooling transients and a poor degree of mixing of the diluted fluid in a PWR vessel. For these types of accidents, mixing is the only mitigative mechanism against high reactivity insertion, which can lead even to re-criticality of the scrammed reactor. For boron dilution
transients, the diluted slug transport and mixing in the vessel after a RCP start-up is the most significant case. The transport of low borated water into the reactor core during re-start of natural circulation after a LOCA might also lead to serious consequences with respect to reactor safety. Early application of system codes indicated large uncertainties. Several experimental programs on boron dilution have been carried out: BORABORA tests in France (EDF), Vattenfall test facility in Sweden, ROCOM test facility in Germany (FZR), UMCP tests in USA, which was used for the International Standard problem 43. A few additional experiments were made available from a Russian VVER-1000 reactor mock-up. All these facilities represent different types of PWR using typically 1/5 scale ratio and tests are performed at atmospheric pressure. The PKL test facility in Germany was used to investigate accumulation and transport to the RPV of slugs of deborated water during small-break and mid-loop operation transients.

Steam line Break
Mixing inside the RPV is also of importance for the analysis of overcooling transients like steam line breaks. In the case of a Main Steam Line Break (MSLB), the primary circuit water is cooled down rapidly in the affected loop due to heat release to the break. The reactivity insertion due to overcooling (or deboration) depends strongly on the distribution of the temperature (resp. boron concentration) at the core inlet, which is affected by mixing. The use of CFD codes allows to increase the knowledge of PWR vessel thermal hydraulics mixing. A comprehensive validation of CFD codes for turbulent mixing problems was performed within the above mentioned FLOMIX-R project. A set of well defined experiments on slug mixing during the start-up of the first pump has been performed at the Rossendorf ROCOM mixing model and the Vattenfall test facility. Within FLOMIX-R, studies have been performed also on mixing under steady-state flow conditions, with all main coolant pumps running or some of them switched off. Corresponding experiments have been performed at ROCOM, but also data from measurements at real plants (NPP Paks in Hungary, VVER-440) have been made available. The data were used to validate mixing matrices calculated with CFD codes. These matrices describe the coolant temperature distribution at the core inlet, when the cold leg temperature in one of the loops is perturbed. They can be applied for MSLB analysis, because the overcooling can be considered as a quasi-stationary process.

Pressurised Thermal Shock
Injection of cold water into the primary circuit of a PWR may produce thermal loads in welds and cracks which are capable to threaten the integrity of the pressure Vessel. Other components of the primary circuit (valves, junctions, elbows, …) may also be affected by thermal shocks, which is a main concern of the ECORA project. To extend the original life of nuclear plants for up to 40 years or more, studies have to be made to assess the integrity of some of these components. A “conservative” approach including high conception margins is also possible but may be a very expensive solution. The refined modelling of the transient thermal loading is then necessary to give realistic conditions to mechanical analysis.

In thermal shock analysis, besides of two-phase flow problems (ECC injection jet mixing, stratified flow in the cold leg), one-phase mixing problems are also of importance. This is mixing of water streams with different temperature and, therefore, different densities in the downcomer. The CFD simulation of buoyancy driven mixing is still a challenge, because advanced turbulence models taking into account the additional shear stress generation by buoyancy forces are necessary. This might be Reynolds stress models, Large Eddy or Detached Eddy simulation. In the project FLOMIX-R, experiments on buoyancy driven mixing performed at the Rossendorf ROCOM mixing test facility (Germany) and at the Fortum PTS facility (Finland) have been performed and calculated with CFD. While the qualitative behaviour of the mixing process can be well described by the codes, significant differences still exist in local parameters (temperature at a certain position e.g. near the RPV wall).

Thermal fatigue
Other applications can also illustrate the benefit of using of CFD codes in order to assess specific types of flows in PWR reactors such as temperature fluctuations inducing thermal fatigue.
For example, a configuration, related to a mixing zone in a PWR reactor, consists in a T junction upstream of a bend in which hot and cold fluid interact, creating temperature fluctuations that might result in thermal fatigue. The use of the thermal coupling between a CFD Code with a solid thermal code gives access to the instantaneous temperature field inside the fluid and the solid. In order to capture fluctuations, Large Eddy simulation may be required for such a problem. Some examples of use of CFD for thermal fatigue analysis may be found in (NEA/CSNI/R(98)8, 1998).

The “Induced Break” issue

During a high pressure severe accident (for example: due to station blackout or loss of secondary feed water), the core is uncovered, heat is transported by steam to the steam generator tubes. Due to a liquid slug in the loop seals a counter current natural circulation takes place in the hot leg and steam generator with direct and reverse circulation in different SG tubes as experimentally observed. The scenario may lead to a failure by thermo-mechanical loads in the primary coolant loop. The flow field and heat transfer details determine whether the failure occurs within the containment, in the reactor coolant piping, in the reactor vessel, or in the steam generator tubing with a leak path out of containment. The key parameters is the mixing within the steam generator inlet plenum (hot steam coming from the core and "cold" steam from the reverse flow in the SG tubes), the intensity of the thermal stratification in the hot leg, and the quantification of direct and reverse flow in SG tubes.

Hydrogen distribution in containment

During a Loss Of Coolant Accidents (LOCA) in a Nuclear Power Plant the Containment pressures due to steam-water discharge from the primary or secondary circuit. Additionally, for certain scenarios hydrogen is generated in the reactor core, and subsequently released into the containment. The gas mixture in the containment can become flammable, and several burning modes of hydrogen can occur including deflagration and detonation. The resulting pressure loads challenge the containment integrity, with a risk of radioactive exposure to the environment. For both the DBA and BDBA conditions above, the transport and mixing of heat, mass, as well as non-condensable gases such as air and hydrogen will affect the containment response. Detailed knowledge of containment thermal hydraulics is necessary to ensure the effectiveness of hydrogen mitigation methods. Containments have very large volumes and multi-compartments. A too coarse nodalisation will not only lose resolution, but will smear the temperature, concentration and velocity gradients through numerical diffusion, thereby leading to an over-homogenization of the hydrogen distribution. Time discretisation is also an important issue, as accident transients must be simulated over several hours, or even days, of physical time. From a physical point of view, the flow model must also take into account condensation (in the bulk or at the wall or due to spay cooling), together with heat transfer to the structures. Condensation models are not standard in CFD codes. An additional, and significant, difficulty in the application of CFD to hydrogen distribution problems relates to the way in which reactor systems, such as recombiners, spray systems, sumps, etc., are taken into account. CFD simulations without such system/component models will not be representative of realistic accident scenarios in nuclear reactor containments.

Hot-leg temperature heterogeneity

Another application deals with the prediction of the flow inside the Upper Plenum and the evaluation of the uncertainties associated with the measurements of temperature into hot legs of a PWR, which is used in a heat balance of the primary coolant flow. One of the purpose is to determine the degree of mixing and the resulting non homogeneous temperature field inside the hot legs.

Other applications

CFD is also used in other components of the primary cooling system such as the main coolant pump where it can help to avoid cavitation erosion in every operating conditions. Other studies
concern temperature fluctuations within the pump thermal barrier and around the shaft, which, together with high radial loadings on the impeller, can lead to shaft crackings.

1.2 How to adjust the ECORA BPG to large scale and reactor problems

Although CFD is being used extensively both inside and outside the nuclear community, still the credibility of many CFD simulations is being discussed. Part of the discussion revolves around the physical difficulties of modelling the effect of, for instance, turbulence. However, another part of the discussion relates to the accuracy of the numerical discretisation in CFD simulations. Difficulties that still exist in this area are amply demonstrated by the many CFD validation exercises involving blind test cases, where only sufficient information is made available to allow a CFD model to be set up and run, but the full test results are not available. The results of such exercises can be highly user-dependent even when the same software and models are used. The ERCOFTAC special interest group on “Quality and Trust in Industrial CFD” has identified that the use of BPG (Casey and Wintergerste, 2000) would reduce these errors and enhance the credibility of CFD. The ECORA project has embraced this notion and, with the ERCOFTAC BPG as a basis, a BPG for nuclear safety applications was created at the beginning of the project (Menter 2002). The emphasis of the ECORA BPG is on validation, which basically means that small scale simulations are performed and, by comparison with experimental data, the extent to which the model accurately represents reality is assessed. In this chapter the experience gained by using the BPG, for both small scale and large scale problems, will be summarised and recommendations for adjusting them will be given.

In the BPG, the following potential sources for errors or uncertainties are defined:

- Numerical errors; difference between the exact equations and the discretised equations (Spatial and temporal discretisation error, iteration error);
- Model errors; error in the applied models, e.g. turbulence models;
- Application uncertainties; lack of information of the application, e.g. boundary condition or details of the geometry;
- User errors; inadequate use of the CFD code by the user, e.g. oversimplification of the problem;
- Software errors; any inconsistency in the software package, e.g. coding errors.

In order to be able to determine the model error separately, which is required for meaningful subsequent model improvement, the other errors have to be minimised. Within the ECORA project this has been done for the following single-phase cases (the two-phase cases will be discussed in a separate chapter):

- PTS Validation 1: Jet impingement with heat transfer (Egorov et. al. 2004);
- PTS Demonstration 1: Upper Plenum Test Facility (UPTF) liquid-liquid mixing (Willemse et. al. 2004);
- Containment Demonstration: Analysis of two PANDA tests (Andreani et. al. 2004)

In the following sections the actual way the BPG were used and reported for these different cases will be described after which recommendations for adjusting the BPG will be given.

1.2.1 PTS Validation 1: Jet impingement with heat transfer

In this validation, case steady-state heat transfer is predicted for a turbulent air jet impinging orthogonally on a large heated plane surface. The computations are performed in a 2D axi-symmetrical geometry.

The iteration error is minimised by applying a convergence criterion, which show no variation of the target variable as it is plotted as a function of the remaining residual. The spatial discretisation error is determined by performing the computations on 5(!) 2x2 successively refined
meshes, for both 1st and 2nd order discretisation schemes. Finally an uncertainty analysis has been performed, by changing the location of the upper boundary, to make sure that the boundary does not disturb the entrainment of the jet. Summarising, application of the ECORA BPG was successfully performed for this validation case.

1.2.2 PTS Demonstration 1: UPTF liquid-liquid mixing

In this large scale demonstration case the transient mixing of cold water injected in, an initially hot water filled, cold leg and downcomer of the Upper Plenum Test Facility (UPTF) is studied. Since this concerns a complex geometry with many internals present in the lower plenum of the reactor vessel, initial calculations were performed by omitting these internals and assuming a porous medium. However, this leads to large unphysical circumferential oscillations. Therefore, in a subsequent calculation, the internals in the lower plenum were modelled in detail, which improved the simulation significantly. So, the geometrical assumptions were tested. The time discretisation error is investigated by reduction of the applied time step; a time step with a Courant number of about one has been used to obtain optimal results for resolving the cold front accurately. Notice that an implicit scheme was used, and this Courant limit is not required for stability of the solution, but for accuracy reasons only. Determination of the spatial discretisation error turned out to be the most difficult for this large scale case. Global mesh refinements turned out to be impossible, since the initial mesh, based on the mesh requirements of the BPG for e.g. y+ values, was already 2.1 million cells. An attempt was made, however, to check the quality of the solution in different parts of the geometry by comparing the first and second order solutions. This showed that the mesh in the cold leg was sufficiently fine, but the mesh in the downcomer had to be refined. Moreover experience indicates that it is particularly important to ensure that the downcomer region around the cold leg nozzle is modelled in sufficient detail. Care is needed to ensure that the geometry in this region is modelled very accurately and finer mesh is required here than elsewhere in the downcomer. It should be noted, that such a division in different parts of a geometry requires that the physics occurring in these different parts do not effect each other substantially. Since it is difficult to prove this beforehand, such an approach should be used with care. Finally, the model error was investigated by using both the SST and k-ε turbulence model, which lead to practically the same results when buoyancy dependent turbulence production/destruction terms are added in both models. Summarising, application of the BPG has been successful for the majority of the guidelines. However, since the initial mesh used for this large demonstration case already contained over 2 million cells, it was not possible to obtain a solution on ‘three (or more) grids using the same topology (or, for unstructured meshes, a uniform refinement over all cells’ as stated in the BPG. In order to make the BPG also applicable to large scale problems, the requirements have to be made less strict in this respect. A suggestion is to add the following guideline to the BPG: When global mesh refinement is not possible, perform calculations with first and second order discretisation on the same mesh, or on multiple meshes with local grid-refinement in areas which are the most sensitive to solution change.

1.2.3 Containment Demonstration: Analysis of two PANDA tests

The results of two experiments on PANDA performed in the SETH project (including basic experiments on gas mixing in a twin-vessel geometry, driven by fluid injection in one of the vessels) will be made available to the ECORA project, and the data will be compared with the pre-test simulations with various CFD codes. The tests characteristics are: low-elevation/low-momentum horizontal injections in one vessel for two different injection velocities, constant pressure, no condensation, and venting (outlet) from the second vessel.
As one of the main goals of the work is to explore the application of the BPG to the simulation of the containment flows that are investigated in the PANDA experiments, an appraisal of the computational overhead implied by the approach had to be obtained in order to set realistic goals. A scoping exercise with a simplified geometry (one vessel only) and boundary conditions similar to those of one of the SETH tests was thus defined. The participants were asked to perform calculations with two meshes for a short transient time (50 s) and estimate the computing time necessary to run the full transient in the actual geometry. The distributions of temperature, concentration and velocity on selected lines were chosen as “target” variables. Following (partly) the BPG, numerical errors were quantified by grid size variations and by comparison of results for first and second order discretisation schemes. Calculations have been performed with CFX-4, FLUENT and TONUS. Additionally, simulations with the GOTHIC code (not among the codes evaluated in the ECORA project) on a coarse mesh were also carried-out.

The main result of the exercise was a realistic estimation of the difficulties to simulate the selected PANDA tests in relation to convergence and running time. This information lead to the conclusion that the BPG will be applied to short time simulations of the flow structure (buoyant jet) in the vessel where the injection takes place, whereas the long-term propagation of the stratification front in the connected vessel will only be simulated for one test using a “practical” mesh.

This two-step approach (application of the BPG to the initial phase of a long transient only) can be useful for the analysis of transients where a slow and “monotonic” evolution of thermal-hydraulic conditions can be expected as a result of approximately constant boundary conditions. For such transients (e.g., for the long-term cooling of passive systems), the verification of the small sensitivity of the initial simulation to the mesh could improve confidence in the results of the simulation obtained for the entire transient with a coarse-mesh model, the only one that can be afforded.

Since the PANDA experimental data was not available before the calculations, these may be considered as blind calculations or demonstration tests for containment-type calculations, with the additional simplicity that no steam condensation is modelled. Still, many features of the calculations (three-dimensional geometry, long-duration transients) make the PANDA benchmark problems interesting from the point of view of testing the capabilities of CFD codes, and in particular, testing the applicability of Best Practice Guidelines to containment calculations.

Conclusions concerning the validity of different choices of turbulence models cannot be drawn before the experimental data availability but their effect can be assessed in terms of smearing of certain target quantities. As with other test cases performed in the ECORA project, the effect of time-step and spatial discretisation can also be assessed by examining certain profiles of target variables.

Before drawing general conclusions about these demonstration tests, experience with two codes are summarized here below.

**Calculations by CEA using the TONUS containment Code**

A certain number of difficulties were encountered by CEA when modelling the PANDA tests. The first difficulty was to generate a three-dimensional mesh, taking into account the particular features of the facility indicated in the report D12 (reference): protrusions, inlet pipe and exit pipe, the latter having a very small diameter. Although the TONUS code uses unstructured meshes, an automatic mesh generation using tetrahedrons would have led to an unsatisfactory mesh. Thus, it was preferred to generate carefully a mesh of hybrid elements (hexahedrons, prisms and tetrahedrons) with particular care near the inlet, outlet and connecting pipe regions. This was quite expensive, but necessary to limit discretisation errors on distorted meshes. From the point of view of mesh and time-step sensitivity, it was not possible to achieve mesh and time-step-independent solutions – even though calculations (using a 3rd order spatial accurate scheme, but only 1st order in time) on three different types of mesh, and using two time steps were performed. Still, the sensitivity of the solution to the mesh spacing and the time-step was observed. Only solutions using the mixing length turbulence model were produced due to lack of time, and no calculation using a k-ε model was performed, so that model error could not be assessed.
Finally, in terms of ability to calculate long transients, CEA managed to calculate the full test 17 up to the required 2000s of physical time-step, using time-steps of the order of 0.2s, on a single-processor LINUX PC. The calculation took several days, but which is still reasonable.

In summary, the BPG could not be applied strictly, but insight into the sensitivity of the numerical solutions to some parameters could be assessed. For application to realistic containment situations (even larger scale, condensation and heat transfer to the structures, and large uncertainties in the initial and boundary conditions), CEA would recommend developing a method to identify the importance of the different parameters on some target variables, to reduce the number of sensitivity studies which are very costly for large-scale applications. The Design of Experiments methodology is such a method, and has been applied by CEA in the CFD analysis of the ISP47 MISTRA Phase A test.

**Calculation by GRS with the CFX code**

At GRS a similar experience was made. Although both tests (T17 and T9) were run for almost all cases (Heitsch, 2004), for a practical case it would take too long before the final solution is available. Three meshes were set-up. They use 147272, 335471 and 817436 fluid cells in a mixed structured-unstructured approach. These meshes are considered to be still too coarse to produce mesh independent results. No major problems were encountered during the simulations.

The mixing problem dealt with in the two PANDA experiments is a simple mixing process without all other phenomena usually found in a real containment scenario. Such a scenario would include conjugate heat transfer, condensation and probably chemical reactions or combustion. Therefore apart from the goal of applying BPG to full extent or in an “ideal” way, each partner involved in the blind simulations, reduced the use of BPG to comply with the given deadlines. This could probably be done in a more systematic way by a ranking of the individual steps within the BPG for a given class of problems. In the actual case the turbulence model and the mesh had the focus but imposed already too much computational effort.

**Conclusions on ECORA BPG application to PANDA tests**

The pre-test analysis of the two tests produced a number of useful indications with respect to the applicability of the BPG for containment applications:

- **Even meshing of rather simple geometries (such as the two vessels of PANDA) is a highly time-consuming job when using either a fully structured mesh of adequate quality or a hybrid mesh.** Most users adopted hybrid meshes, as a fully unstructured mesh would lead to inaccurate results. Specific features of the PANDA facility required some level of detail in certain regions, and the issue of the best meshing strategy still has to be considered in future exercises. In the case of fully structured mesh (as necessary, e.g., for the CFX-4 code) the sensitivity of the convergence of certain solvers to the details of the mesh used seems to be very high, so that the CPU time may be highly dependent on the mesh and the analysis may require a number of attempts. Any conclusion or recommendation about the use of hybrid meshes or a structured would be premature since the analysis of the results is still in progress.

- **Considering the large computation times required even for the short time simulation, the BPG had to be relaxed also for the initial period.** In particular, the refinement of the mesh in two steps (coarse, intermediate, fine) was based on expert judgement rather than on global refinement in the three directions. In fact, global refinement would have been impossible, as the basic mesh must include 10^5 cells. The finest mesh for mainly unstructured meshes would have thus consisted of more than 6 million cells. Moreover the sensitivity to the time step was required for the intermediate mesh only.

- **BPG would require the use of substantially different turbulence modelling for estimating the model error.** Again because of the CPU cost associated with the application of higher-order models, the use two models, that produced sufficiently different results for the same class of flows, may be preferred.

- **Calculations have been produced with the CFX-4, CFX-5, FLUENT and TONUS codes.**
The number of sensitivity studies (and the degree of compliance with the BPG) varied for the various codes and users, depending on the difficulties encountered in the meshing and resources available. One set of simulations (NRI with FLUENT) completely fulfilled the reduced requirements discussed above, and these results can be taken as an example of what can be achieved at present. In this analysis, three meshes were used and three simulations with various time steps were executed for the initial period. The finest mesh comprised four times more cells than the coarsest one. The long transient calculation with the intermediate mesh (480,000 cells) required 6 weeks on two processors. This analysis shows that application of reduced BPG and a two-step approach for long transients can be afforded, although was not reasonable for all applications.

The sensitivity of the results to the mesh used was reported by several users to be quite high, although is not clear at present to what extent this may depend on the details of the mesh in specific regions, specially in the injection region. The results of NRI, however, which also include regions with unstructured mesh, show an encouraging picture of the results that can be obtained. Although results obtained with three meshes could not reach a converged solution in a rigorous sense (cell size was not successively reduced by two in all directions in the regions with hexahedral cells), the RMS of the difference of the results for the target variables was of the order of 0.02 or less. For all practical purposes, a solution with such a low sensitivity to the mesh refinement can be considered as converged.

Predictions with the various models and codes produced an unexpected broad dispersion band. Although additional analysis will be required to clarify the effect of a number of choices in relation to boundary and initial conditions and the effect of the mesh (not always investigated in a systematic manner), it is clear that the simulation of mixing of gases in large vessels is highly sensitive to the turbulence model, even when using only variants of the k-ε model. In particular large differences were obtained by GRS with the standard k-ε model and with the SST model, and the simulations of NRI showed that the RMS of the difference of the results with the standard and the RNG model were much higher (around 10%) than the variations due to the mesh, which showed that the model error could be quantified as the numerical error was minimised.

The observations above (based on a preliminary analysis of the simulation results only) lead to a number of recommendations that are proposed in the following section.

1.2.4 Recommendations

All the BPG could be applied for the small scale (2D) validation case in the project. Also, the strict use of the majority of the BPG for a large scale (3D) demonstration case has given insight in the errors in this simulation. However, obtaining a solution on three successively refined grids turned out to be impossible, due to the expected computational demand of these calculations. So, the BPG have to be adapted adjusted and extended to be applicable for large scale problems and reactor problems. This is a general recommendation that can be drawn from first attempts to apply BPG to large scale problems such as PANDA or UPTF at the end of ECORA project. Such an extension of the BPG requires further work and cannot be established during ECORA project but first suggestions emerged from ECORA members:

- When global mesh refinement is not possible, it is recommended to perform calculations with first and second order discretisation on the same mesh,
- It is also recommended to perform calculations on multiple meshes with local grid-refinement in areas which are the most sensitive to solution change although this may be difficult for transient flows where the areas of the flow with high gradients can change in time.
- For large-scale problems with multiple parameters, it might also be recommended to incorporate into the BPG a “Design of Experiments” methodology to limit the cost of evaluating the different types of errors in CFD computations.
For the specific class of flows relevant to containment analysis as studied in the present project, the application of the BPG (though relaxed as regards the number of sensitivity studies required) seems to be feasible for the initial period of the transient. A two-step approach can be proposed, namely the application of the BPG to a portion of the transient to “qualify” the finest practical mesh that can be afforded for the complete transient. The definition of “practical mesh” will obviously depend on the available computer resources and company policies in assigning priorities to the various applications. In the framework of this approach, the use of three successively refined mesh is affordable, and sufficiently converged results can be achieved, although this can imply a large effort in optimising the mesh. Finally, the use of advanced (and highly computing intensive) turbulence models may be not necessary for defining the range of uncertainty due to the choice of the turbulence model. Previous experience of similar flows should help to select the models for estimating model errors. It should however be noted, that the range of applications, where the strict application of BPG is feasible, will continuously expand with the ever increasing computing power. Recommendations for reduced BP procedures should therefore be considered as a temporary solution, which will gradually converge back to the strict application of the procedures. For three-dimensional single-phase flows, this should be possible within the next decade, at least for validation studies.

2 USE OF TWO-PHASE FLOW CFD

The EC project EUROFASTNET, which was a pre-runner to ECORA, has identified industrial needs for three-dimensional simulation of nuclear reactor thermo-hydraulics. These include safety, performance, design, availability and increase of life span of nuclear reactors (Bestion et al., 2002). The requirements with the highest industrial priority are fuel performance, fluid-structure interaction, thermal shocks due to safety injection and stratification in circuits. As a consequence, the ‘Extension of CFD Codes to Two-Phase Safety Problems’ has become the subject of Writing Groups on CFD Issues, which has been established by the OECD/NEA. The report produced by this Writing Group (Bestion et al., 2004) covers a wide range of NRS problems in pressurized water reactors (PWR), boiling water reactors (BWR), steam generators, heat exchangers, containment flows and components with three-dimensional structures like spacer grids. High priority is given to critical heat flux conditions in the core, to two-phase pressurized thermal shocks (PTS), and to thermal fatigue and stratification in the primary system of PWRs. The reports of Bestion et al. (2002) and Bestion et al. (2004) embrace all aspects of two-phase flow CFD in NRS. In the ASTAR project, some test cases of interest for two-phase flow modelling were investigated, and BPG were also applied to produce better quality solutions (de Chadard, 2003) and (Romstedt, 2003). In the EC ORA project, only a subset of the cases described by Bestion et al. (2004) was investigated, namely the CFD simulation of flows in the primary system and containment of PWRs. Two-phase flow phenomena were studied for PTS-relevant flow conditions during the injection of emergency core cooling water into the cold leg of PWRs. The following discussion will be focussed on these topics.

2.1 Nuclear Reactor Safety Problems where CFD is recommended

Two-phase CFD tools are far less mature than single phase tools. However, it is recommended to further develop such tools for a number of nuclear safety issues. Due to the lower maturity, the activity should include:

- Identification of the relevant basic phenomena which need to be modelled for a given application
- Assessment of the model including verification, validation and demonstration tests
- Definition of new R&D work for a more detailed validation, for a better numerical efficiency, and a better accuracy and reliability of predictions
Such a process has been applied within ECORA to the two-phase PTS scenario as summarized here below.

In a typical PTS scenario, cold water is injected into the cold leg of a PWR during the refill phase of a LOCA. The injected water mixes with the hot fluid present in the cold leg. Depending on the size of the break, either single- or two-phase conditions prevail between the injection nozzle and the downcomer. There may be stratification of cold water on the bottom of the cold leg with counter-current flow of hot water or steam on top of the cold-water layer. Condensation phenomena take place at the free surfaces of the cooling water jet and of the stratified flow. These depend strongly on the turbulence in the fluid. As a consequence of the emergency cooling water injection, a stream of cold water penetrates into the downcomer. The path and characteristics of this cold-water jet depends on the flow conditions and on the detailed cold leg, downcomer geometry. Thermal shocks can occur on the reactor vessel walls due to the thermal and mechanical stresses arising from the rapid temperature and pressure changes at the cold-water jet edge. This, in turn, can lead to mechanical failure of the walls.

The dominant fluid and heat transfer phenomena involved in the two-phase flow scenarios described above are:

- Impingement of two-phase flow jets
- Impinging jet heat transfer
- Turbulent mixing of momentum and energy in and downstream of the impingement zone
- Stratified two-phase flow (or free surface flow) within ducts
- Phase change, like condensation at the steam-water interface

These phenomena were investigated in ECORA. The respective test cases which were calculated in ECORA are documented in the Deliverable D05a (Scheuerer, 2003). The ECORA test cases were subdivided into verification, validation and demonstration tests. A detailed description of the results is given in the Deliverable D06 (Egorov, 2004).

The validation tests included jet impingement with heat transfer, water jet impingement in an air environment, and contact condensation in a stratified steam-water flow. It was shown that turbulence model formulations based on the $\omega$-length scale equation are well suited for the simulation of impinging jet flows. The characteristics of the free surface water jet flow were also adequately represented by the free surface flow models.

The simulations of the contact condensation have shown that standard interfacial mass transfer models are not sufficient to predict this phenomenon accurately. However, an improved condensation model, implemented in CFX-5, which identifies the free surface by calculating the gradient of the volume fraction, and damps the turbulence at such identified free surface shows satisfactory agreement with data.

In summary, the calculations in ECORA have shown a satisfactory performance of the employed CFD codes for single-phase flow problems, and for two-phase flow problems with single dominant interface morphology. This includes free surface flows, or bubbly flows. However, for cases with more than one morphology, for instance for a jet impinging on free surface flow with bubble entrainment or for the transition of bubble to churn flow, the available two-phase models show poor results and need to be improved. The same is true for multi-phase flows with heat transfer and mass transfer at the interface. For the latter, the numerical schemes and the physical models need enhancements. A prerequisite for these model improvements is, however, the provision of adequate experimental data to develop, calibrate and validate these models.

The test calculations in ECORA have also shown that the calculation times for typical PWR assemblies (cold legs, downcomer, lower plenum, core, and hot legs, see Egorov, 2004) are still very large (order of weeks on current parallel machines). Improvements in the numerical methods, and advancements like error-based grid and time-step adaptation, are therefore necessary to make CFD a tool for comparing and assessing different scenarios in NRS.

The ECORA project has made a contribution to the assessment of the state-of-the-art of two-phase flow CFD in NRS. Also, a number of useful model improvements have been made in ECORA. It is, however, still difficult to ‘recommend NRS problems’ for CFD. If a problem is suitable or not depends on the accuracy expectations and on the computational investments one
is willing to make. As said above, many two-phase flow problems with single morphologies (free surfaces, bubble flows, droplet flows, water jets, …) can be predicted quite well. However, there are still large uncertainties for flows with more than one morphology, and for flows with mass transfer at the interface (condensation, boiling, cavitation). Also, calculation times for these kinds of flows may become prohibitive. Therefore, while many of the flows described by Bestion et al. (2004, Table I) can now be tackled from a research and development perspective, the available CFD codes are not yet sufficiently mature for a day-to-day industrial application. Independent of the test case and the problem at hand, ECORA has shown that the application of the Best Practice Guidelines developed as part of the project, can substantially reduce uncertainties, lead to more valid conclusions about model performance vs. performance of the numerical method, and can therefore help to accelerate development and progress of three-dimensional CFD in NRS. 

A similar approach should be followed for other important issues with two-phase flow such as:

- DNB, dry-out and CHF investigations
- Direct contact condensation: ECCS injection or steam discharge in a pool
- Condensation induced water-hammer
- Pool heat exchangers: thermal stratification and mixing problems
- Corrosion, erosion and deposition
- Two-phase flow in valves, safety valves
- Flow oscillations in BWRs
- Steam generator tube vibration
- Pipe Flow with Cavitation

### 2.2 How to adjust the ECORA BPG to reactor relevant two-phase problems

Best Practice Guidelines have been specified. They will form the framework for the CFD simulations. The purpose of the present section is to provide some additional comments to the BPG, concerning their application to the real situations encountered in two-phase flow modelling.

#### 2.2.1 Verification Test-Cases for Two-Phase Flow Codes

#### 2.2.1.1 Objectives of the verification

The purpose of verification test cases can be to ensure the correct implementation of all numerical methods or all closure laws in a CFD method. Two-phase flow codes have initially been developed for industrial applications with large geometrical scales. In such large-scale flows, the structures of dispersed phases like bubbles or droplets cannot be recognised individually. In cases where the bubble sizes are significantly smaller than the sizes of the numerical cells, their behaviour must be schematically represented via statistical closure laws. Two-fluid equations are then averaged in a similar way to the RANS approach in single phase flows. Mass, momentum and energy equations are solved for each phase. The presence of individual phases in the domain are indicated by means of statistically averaged volume fractions. The transport equations solved in multi-phase flow codes are much more complex than those of single-phase flow codes. The potential of numerical methods in multi-phase codes is thus difficult to estimate a priori and the analysis of many computational results is needed to evaluate the numerical capabilities. The same is true concerning the validation of the various physical closure laws.
2.2.1.2 Content of the verification

There is a need to perform verifications calculations in as simple as possible situations, but nevertheless related to nuclear safety analysis with single-effect cases, in which one physical phenomenon is dominant. These cases must however be realistic with regard to the application of two-phase flow codes. The best verification data would be analytical solutions for simple cases, which allow testing all relevant implementation aspects of a CFD code. As analytical solutions are practically never available in two-phase flow, simple experimental test cases are often used instead.

In a second step, on the basis of the experience in the computational analysis of multi-phase flows, heuristic criteria can be defined to be fulfilled by the computational results. The criteria deal with the general coherence of algorithms with the physics of phenomena, the robustness of the algorithms with regard to pressure variations and residual phases treatment, accuracy with regard to numerical diffusion (and dispersion), mass and energy balances. The proposed approach allows separating numerical and physical effects on the results. Consequently, following this approach helps users to improve the knowledge about the codes, and helps to avoid undesirable ‘black box’ effects, when ‘unexpected’ results are obtained in practical situations. In such cases, one can refer to previous studies of analytical test cases, to point out potential sources of uncertainties like balance errors, numerical diffusion or dispersion. The tests cases chosen in the ECORA project are the Oscillating Manometer and the Sloshing (Pigny et al., 2003).

2.2.1.3 Requirements about verification

The only requirement for verification data is that they must allow to check the correct implementation of the code and/or the models. This requires other sources of information (analytical solution, experimental data) about the test case to allow comparison. Strictly speaking, a perfect agreement with the data is not required, but the differences between the simulations and the data must remain in a domain prescribed by criteria.

The test series for model or numerical verification must be diverse enough to check all aspects of the implementation and to allow the examination of all criteria defined.

Software verification for physical models should be carried out in the same environment that is available to the end-user.

Verification cases should be selected before the model is implemented. They must be considered an integral part of the model implementation.

2.2.2 Control of errors in two-phase flows CFD simulations

2.2.2.1 Solution Error

The most relevant errors from a practical standpoint are solution errors. They are the difference between the exact solution of the equations of the model and the numerical solution. They include the errors thereafter described.

2.2.2.2 Spatial Discretisation Error

Some of the spatial discretisation errors are the result of truncation errors in numerical approximations of the derivatives or integrals. The truncation error can be obtained by a Taylor series expansion of the numerical solution for the different terms of the discretised equations:
\[ f_{\text{numerical}} = f_{\text{exact}} + \sum_{i=1}^{\infty} c_i f^{(i)} \Delta \]  

where \( f^{(i)} \) is the \( i \)-th derivative of the exact solution at a given location.

An example is a central difference for a spatial derivative:

\[
\frac{\partial f}{\partial x} \approx \frac{f_{i+1} - f_{i-1}}{x_{i+1} - x_{i-1}} = \frac{(f_{\text{exact}} + f^{(1)} \Delta x + c_2 f^{(2)} \Delta x^2 + c_3 f^{(3)} \Delta x^3 + \text{HOT}) - (f_{\text{exact}} - f^{(1)} \Delta x + c_2 f^{(2)} \Delta x^2 - c_3 f^{(3)} \Delta x^3 + \text{HOT})}{2\Delta x} = f^{(1)} + O(\Delta x^2)
\]

This formulation has a truncation error of order 2 and is therefore second order accurate. The overall truncation error order of the spatial discretisation scheme is determined by the lowest order truncation error after all terms have been discretised.

In the \( O(\Delta x^2) \) term the leading term is proportional to \( f^{(3)} \Delta x^2 \). First order upwind differencing of the convective terms yields truncation errors \( O(\Delta x) \) with leading term proportional to \( f^{(2)} \Delta x \). This term then contributes artificially to the diffusion (numerical diffusion). Such schemes also enhance the dissipation property of the numerical algorithm, (Ferziger, Peric, 1996).

From a practical standpoint, for a first order method the error is divided by 2 by a doubling of the grid resolution in each direction. For a second order method, it is divided by 4 for the same grid refinement.

In CFD codes, the effect of truncation has an important influence on the transport term. Generally speaking, even order truncation terms lead to numerical diffusion and odd order ones to numerical dispersion. The order of the space differencing scheme is thus 1 or 2 in most CFD codes. Notice that when a second order scheme is used, the first order scheme can be used locally in space and time, to smooth dispersive effects, connected to high pressure gradients.

In two-phase systems, the truncation error on the algebraic source terms can be of prime importance. Interfacial transfers of heat, mass and momentum are modelled by strongly non-linear stiff source terms in some flow conditions. Most codes use zero order centred discretization of these terms, which results in a truncation error of order 1. Nevertheless, one should not forget that the numerical diffusion depends not only on the discretisation of transport terms but also on the time discretisation. The real order of the numerical method can be obtained by the slope of the logarithm of the error for a target variable, plotted versus the logarithm of the cell size, when the cell size tends to zero. In the same way the time discretisation of stiff source terms is of prime importance when the associated relaxation time constants are very small.

Estimating truncation error and trying to reach converged meshing encounters many difficulties in two-phase CFD:

- Choice of the target variable: many physical parameters can be used as target variables to measure the error. For instance, in the verification test case “sloshing” (Pigny, 2003), the height of the second peak is the major experimental result, whereas the water column is thin, and involves a low amount of water. It is not obvious how important is the accuracy on this parameter.
- Present computational resources only allow two-phase flow calculations with mesh sizes far from the convergence. Thus the numerical diffusion and dispersion can only be estimated via the “engineering judgment” (an important effect of the numerical diffusion, relatively easy to observe, is that it leads to minimize the amplitude of the waves).
• The physical filtering of basic equations due to averaging should be clearly distin-
guished from the numerical filtering due to the discretization. Phenomena which are
larger than the physical filter scale can be simulated with more or less numerical accu-
racy but phenomena which are smaller than the physical filter scale should be clearly
modelled by closure laws. It is a necessary condition for allowing mesh refinements
up to convergence. Another approach could be to benefit from mesh refinement for
simulating smaller scale phenomena (like in the study about the Bonetto and Lahey
experiment ,1993), but this should be clearly identified as an extension of the Large
Eddy Simulation approach to two-phase CFD and the classical mesh convergence is
no more relevant.

In any case, mesh convergence tests is recommended to estimate, even roughly, the truncation
errors. For most cases 3 to 5 different mesh sizes are useful to obtain the information. Generally
the practical order of the numerical methods implemented in the industrial software is, for regu-
lar solutions:

• Between 1+ and 2- for regular and high quality meshes
• Between 0+ and 1 for low quality meshes

Concerning the problem with very strong discontinuities, as for instance shock problems, the
order of the methods rarely exceeds unity. The present analysis shows that the problem of spa-
tial discretisation errors is directly connected to the order of the numerical scheme and to the
need to perform calculations with different mesh sizes.

2.2.2.3 Iteration Error

Iteration error per time step is quantified by plotting results for the target quantities for different
convergence criteria, showing to which extent the computation of the target quantities has be-
come independent of the convergence criterion. from a global point of view, mass and energy
balances can be written over the computational domain, during one time step. The accumulation
of errors over the domain can be followed during the simulation. More over, the global error
over the domain can be time summed over the time simulation, to estimate the global balances.
A macroscopic view of iteration errors can thus be obtained.

The problem of iterative convergence can be seen in different ways, for different numerical
schemes. For a code using a fractional step method, each step corresponds to a particular physi-
cal phenomenon, which is characterized by a specific time scale. This is convenient for the in-
vestigation of situations in which different processes of different characteristic times are in-
olved. A stiff problem can be treated as a sequence of less stiff problems. Severe convergence
criteria have to be chosen for each step. For instance, the approach for the Neptune code is quite
different from the CFX code, in which coupled equations are solved. This involves linearisation
of operators. Consequently, the notion of iterative convergence has not the same meaning for
different numerical schemes.

2.2.2.4 Round-off Error

The round-off error can be estimated by performing calculations in single and in double preci-
sions. Thus one can compare the difference between the results and a reference solution, and
compare the CPU time required for the two calculations.

2.2.3 Concluding remarks

Two-phase CFD is much less mature than single phase CFD. The flows are much more complex
and myriads of basic phenomena may take place at various scales. Thus it is clear that the
physical modelling will have to be improved over a long time period. Fundamental questions related to the averaging or filtering of equations are not yet as clearly formalised as they are for RANS or LES methods in single phase. This makes that the separation between physics and numerics is not always well defined. This lack of maturity is also reflected in the Best Practice Guidelines which cannot be as clearly defined as in single phase flows. One may expect that new ideas about extension of BPG to two-phase will emerge in parallel to the progress in modelling and understanding of two-phase flows. Finalising BPG in two-phase applications will not be achieved in ECORA project, and ECORA strongly recommends further investigations on this topic.

3 REFERENCES

Andreani, M., Heitsch, M., Karppinen, I., Henriksson, M., Paillere, H., Macek, J., Toth, I. “Summary of selected tests and criteria applied to choice of model, mesh and numerical methods”, EVOL-ECORA D12, 2004a


Egorov, Y., Boucker, M., Martin, A., Pigny, S., Scheuerer, M., Willemse, S., “Validation of CFD Codes with PTS-relevant Test Cases”, EVOL-ECORA D06, 2004

Heitsch, M., Simulation of SETH Experiments T9 and T17 with CFX5, ECORA Project Meeting, Petten, 2004

IAEA/NEA workshop on the Use of CFD codes for safety analysis of reactor systems including containment, Pisa, Italy, 11-13 November 2002

IAEA-TECDOC-1163, Heat transport and afterheat removal for gas cooled reactors under accident conditions, 2000

IAEA-TECDOC-1382, Evaluation of high temperature gas cooled reactor performance: benchmark analysis related to initial testing of the HTTR and HTR-10, 2003

Menter, F., “CFD Best Practice Guidelines for CFD Code Validation for Reactor-Safety Applications”, EVOL-ECORA D01, 2002

NEA/CSNI/R(98)8, Proc. Specialist meeting on experience with thermal fatigue in LWR piping caused by mixing and stratification, Paris, France, 8-10 June, 1998


Scheuerer, M., “Selection of PTS-Relevant Test Cases”, EVOL-ECORA-D05a, 2003


Willemsen, S., Scheuerer, M., Martin, A., “Demonstration of CFD Codes with PTS-relevant Large Scale Cases”, EVOL-ECORA D09, 2004