ASSESSMENT OF CFD CODES USED IN NUCLEAR REACTOR SAFETY SIMULATIONS

BRIAN L. SMITH
Thermal Hydraulics Laboratory, Nuclear Energy and Safety Department
Paul Scherrer Institute, CH-5232 Villigen PSI, Switzerland
*Corresponding author. E-mail : brian.smith@psi.ch

Received July 13, 2010

Following a joint OECD/NEA–IAEA–sponsored meeting to define the current role and future perspectives of the application of Computational Fluid Dynamics (CFD) to nuclear reactor safety problems, three Writing Groups were created, under the auspices of the NEA# working group WGAMA*, to produce state-of-the-art reports on different aspects of the subject. The work of the second group, WG2, was to document the existing assessment databases for CFD simulation in the context of Nuclear Reactor Safety (NRS) analysis, to gain a measure of the degree of quality and trust in CFD as a numerical analysis tool, and to take initiatives to extend the existing databases. The group worked over the period of 2003-2007 and produced a final state-of-the-art report. The present paper summarises the material gathered during the study, illustrating the points with a few highlights. A total of 22 safety issues were identified for which the application of CFD was considered to potentially bring real benefits in terms of better understanding and increased safety. A list of the existing databases was drawn up and synthesised, both from the nuclear area and from other parallel, non-nuclear, industrial activities. The gaps in the technology base were also identified and discussed. In order to initiate new ways of bringing experimentalists and numerical analysts together, an international workshop -- CFD4NRS (the first in a series) -- was organised, a new blind benchmark activity was set up based on turbulent mixing in T-junctions, and a Wiki-type web portal was created to offer online access to the material put together by the group giving the reader the opportunity to update and extend the contents to keep the information source topical and dynamic.

KEYWORDS : OECD/NEA; CFD; Verification; Validation; Assessment; Databases; Quality and Trust.

1. INTRODUCTION

The spectacular growth in computer hardware over the last quarter century and the accompanying advances in software development have resulted in the availability of reliable numerical tools for addressing safety issues in Nuclear Power Plants (NPPs). The first step forward was undertaken in the 1970s with the development of system codes using the two-fluid model approach [1], such as RELAP-5 [2], TRAC/TRACE [3], CATHARE [4] and ATHLET [5] for example, for the analysis of primary circuit transients. Other programs, such as GOTHIC [6], GASFLOW [7], MELCOR [8] SCDAP [9] and MAAP [10] have also been written for containment and severe accident analyses, respectively.

The application of Computational Fluid Dynamics (CFD) methods to problems relating to Nuclear Reactor Safety (NRS) is less well developed but is rapidly accelerating. The need to use CFD arises because many traditional reactor system and containment codes are based on a network of 1-D or 0-D volumes. It is evident, however, that the flow in such components as the upper and lower plena, downcomer and core of a Reactor Pressure Vessel (RPV) is strongly three dimensional. Natural circulation, mixing and stratification in containments is also essentially 3-D in nature, and representing such complex flows by pseudo 1-D approximations may not just be oversimplified but could even be misleading, resulting in erroneous judgments being made.

One of the reasons why the application of CFD methods in NRS has been slow to establish itself is that the transient, often two-phase, phenomena associated with accident events are extremely complex. Traditional approaches using system codes have been successful because a very large database of phasic exchange correlations has been built into them. The correlations have been formulated from 1-D special-effects experiments and have been well tested. Data on the exchange of mass, momentum, and energy between phases for 3-D flows are very sparse in

#Nuclear Energy Agency
*Working Group on the Analysis and Management of Accidents

comparison. Thus, although 1-D formulations may restrict the use of system codes in simulations in which there is geometric complexity and 3-D fluid motions, the physical models are well-established and reliable, provided they are used within their specified ranges of validity, and these days they are often run in real time for full reactor transients. In contrast, to use CFD, the physical models will require considerable further development, especially for two-phase applications; also, massively parallel machine architecture is a necessity for real reactor applications.

The use of numerical simulation methods in NRS often has to address regulatory concerns. From this perspective, a common approach to dealing with practical licensing issues is to use simplified modelling, coupled with a conservative approach, to ensure that adequate safety margins are guaranteed. Traditionally, a large number of sensitivity studies are carried out to determine how plant parameters have to be modified in order for the predictions to remain conservative. Sophisticated statistical methods, such as Latin Hypercube Sampling (LHS), have placed this practice on a firm mathematical foundation. However, a key concern is to determine the degree of conservatism needed to compensate for the lack of physics embodied in the simplified models. Information can be obtained from mock-up experiments, but difficult scaling issues have to be faced [11] in order to ensure that the extrapolation of model data to full scale is trustworthy. Moreover, the experiments themselves inevitably involve simplifications, and judging the degree of conservatism associated with introducing the simplifications is itself quite difficult. The only way to ultimately ensure that a conservative approach has been proposed is to increase safety margins, but this practice often places unwelcome constraints on plant efficiency and competitiveness.

The trend being taken by most safety authorities is to gradually replace a conservative approach by a best-estimate methodology, coupled with an uncertainty evaluation [12]. This policy change has already taken place in the context of system analysis with the development of second-generation codes in the 1970s, based on the two-fluid approach as a means of replacing the conservatism of simplified two-phase flow models. The use of CFD codes in NRS may be viewed similarly in regard to having an appropriate numerical tool to analyse certain situations in which there is a strong 3-D flow component.

To gain acceptance in the licensing world, investigations need to be underpinned by a comprehensive validation programme to demonstrate the capability of the technology and to provide results reliable enough to be used in licensing procedures. For single-phase applications, CFD is mature enough to complement existing analysis tools currently employed by the regulatory authorities, and it has the potential to reduce conservatism without compromising safety margins. However, one issue that needs to be resolved is that generally the major commercial CFD vendors do not allow unrestricted access to their source code, a situation which could be unacceptable from a regulatory standpoint. The use of open source software such as OpenFOAM [13] offers a way to circumvent this difficulty. An alternative would be for the authorities to use a well-established CFD code as a cross-check on the safety submissions based on CFD that have been presented to them. One thing is for certain, however, CFD will enter the safety picture in an increasing way in the near future.

2. CSNI[#] ACTION PLAN

The starting point for the activities reported in this article was an *Exploratory Meeting of Experts to Define an Action Plan on the Application of Computational Fluid Dynamics (CFD) Codes to Nuclear Reactor Safety (NRS) Problems*, which was held in Aix-en-Provence, France on 15-16 May, 2002 [14], a meeting jointly sponsored by the IAEA[*] and the OECD/NEA[‡]. This initiative resulted in the formulation of an action plan recommending the creation of three Writing Groups, overseen by the OECD/NEA, and with mandates to perform the following tasks:

- WG1 Provide a set of guidelines for the application of CFD to NRS problems;
- WG2 Evaluate the existing CFD assessment bases, identify any gaps, and initiate activities aimed at broadening the assessment database;
- WG3 Summarise the extensions needed to CFD codes for application to two-phase NRS problems.

The present paper summarises the work undertaken within Writing Group WG2 as a result of this initiative. Work began early in 2003. Teams of experts were assigned to each of the groups, representing the following OECD member countries: the Czech Republic, France, Germany, Italy, Japan, S. Korea, the Netherlands, Norway, Sweden, Switzerland and the USA. A preliminary report was submitted to the OECD/NEA Working Group on the Analysis and Management of Accidents (WGAMA) in September 2004, which scoped the work needed to be carried out to fulfil the WG2 mandate and made recommendations on how to achieve the defined objective. The other groups followed a similar procedure, and in January 2005, all three groups were re-formed to carry out their respective tasks.

The WG2 group concentrated on single-phase

[#]Committee on the Safety of Nuclear Installations
[*]International Atomic Energy Authority
[‡]Organisation for Economic Cooperation and Development, Nuclear Energy Agency

phenomena; two-phase CFD was not yet considered to be of sufficient maturity for a comprehensive assessment basis to be constructed, and the identification of the areas which still need to be developed (the task of WG3) should be undertaken first. It was recognised that, unlike the situation with system and containment codes, the nuclear community was not the primary driving force for the development of commercial CFD software, but could benefit from the validation programmes originating in non-nuclear areas, since often the thermal-hydraulic phenomena were similar.

The remainder of the paper is organised as follows. Section 3 lists those NRS issues identified by the group for which it was considered that the application of CFD would bring real benefits in terms of better predictive capability over traditional lumped-parameter or 1-D approaches. Some highlights are included for illustrative purposes, but the reader is referred to the CSNI report [15] for full details. In Section 4, brief descriptions of the verification, validation and assessment procedures are given, and Section 5 details the assessment bases that have already been established in the non-nuclear domain and discusses their usefulness and relevance to NRS applications. Most CFD codes currently being used for NRS analyses have their own, custom-built assessment bases; these are included in the list. Since many of the phenomena occurring in reactor thermal hydraulics are very similar to basic fluid flow situations appearing in other circumstances, this non-nuclear assessment database is very useful even in the context of nuclear applications.

Table 1. NRS Problems Requiring CFD with/without Coupling to System Codes

	NRS problem	System classification	Incident classification	Single- or multi-phase
1	Erosion, corrosion and deposition	Core, primary and secondary circuits	Operational	Single/Multi
2	Core instability in BWRs	Core	Operational	Multi
3	Transition boiling in BWR/determination of MCPR	Core	Operational	Multi
4	Recriticality in BWRs	Core	BDBA	Multi
5	Reflooding	Core	DBA	Multi
6	Lower plenum debris coolability/melt distribution	Core	BDBA	Multi
7	Boron dilution	Primary circuit	DBA	Single
8	Mixing: stratification/hot-leg heterogeneities	Primary circuit	Operational	Single/Multi
9	Heterogeneous flow distribution (e.g. in SG inlet plenum causing vibrations, HDR expts., etc.)	Primary circuit	Operational	Single
10	BWR/ABWR lower plenum flow	Primary circuit	Operational	Single/Multi
11	Waterhammer condensation	Primary circuit	Operational	Multi
12	PTS (pressurised thermal shock)	Primary circuit	DBA	Single/Multi
13	Pipe break – in-vessel mechanical load	Primary circuit	DBA	Multi
14	Induced break	Primary circuit	DBA	Single
15	Thermal fatigue (e.g. T-junction)	Primary circuit	Operational	Single
16	Hydrogen distribution	Containment	BDBA	Single/Multi
17	Chemical reactions/combustion/detonation	Containment	BDBA	Single/Multi
18	Aerosol deposition/atmospheric transport (source term)	Containment	BDBA	Multi
19	Direct-contact condensation	Containment/ Primary circuit	DBA	Multi
20	Bubble dynamics in suppression pools	Containment	DBA	Multi
21	Behaviour of gas/liquid surfaces	Containment/ Primary circuit	Operational	Multi
22	Special considerations for advanced (including Gas-Cooled) reactors	Containment/ Primary circuit	DBA/BDBA	Single/Multi

DBA – Design Basis Accident; BDBA – Beyond Design Basis (or Severe) Accident; MCPR – Minimum Critical Power Ratio

Nonetheless, databases that have been established with NRS issues specifically in mind represent the most valued data source for the document; these are described in some detail in Section 6. Typical examples are experiments devoted to the boron dilution issue, pressurised thermal shock and thermal fatigue. The technology gaps which need to be closed to make CFD a more trustworthy numerical tool for NRS analyses are listed in Section 7. Section 8 describes the new initiatives taken by the group to broaden the established assessment bases. These include the setting up of a new series of international workshops under the acronym CFD4NRS specifically focussing on the use of CFD in nuclear reactor safety research, the launching of a blind benchmarking activity in the field of thermal fatigue, and the creation of a Wiki-type web portal to store, update and extend the information compiled by the group. Finally, a summing up is given in Section 9.

3. NRS PROBLEMS FOR WHICH CFD COULD BRING REAL BENEFITS

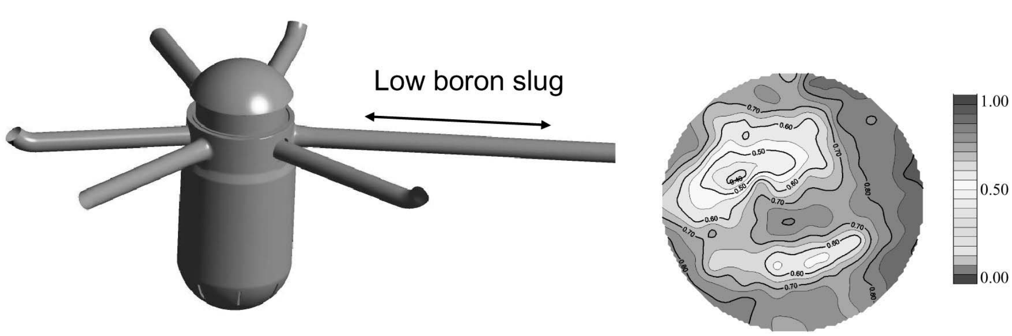
Table 1 lists the NRS problems for which the group considered CFD could bring real benefits in terms of better understanding, quantification, and improved safety estimation. To be included on the list, the information supplied was cast in the following form: (i) what is the relevance to nuclear reactor safety; (ii) why is CFD needed; (iii) what is the current state-of-the-art on the subject; and (iv) what are the perspectives for improvement? Both single and multi-phase problems were identified, though in the latter case only the briefest of descriptions was given, and the details left to be discussed within the framework of Writing Group WG3 [16], in which the modelling extensions that would be necessary for CFD to handle such problems are reported.

The entries on the list do not in any way reflect priorities or degrees of interest in the problem. Prioritising the safety issues was the task of a separate study group [17]. Rather, with some areas of overlap, the safety issues are grouped into problems concerning the reactor core, the primary circuit, or the containment; they are listed in this order. Full details are given in the CSNI document [15], but some salient points are picked up here for illustrative purposes.

3.1 Boron Dilution

Mechanisms have been identified [18], such as SB-LOCA or steam generator tube rupture (SGTR), which could lead to a slug of low borated water being injected through one of the coolant loops into the RPV of a Pressurised Water Reactor (PWR). If the slug arrives at the core without mixing significantly with the streams from the other cold legs, a (local) criticality excursion could ensue. The complete phenomenological model requires two steps: (i) knowledge of the concentration of boron at the core entrance, and (ii) thermal-hydraulics/ neutronics calculations for the core region. The first step (covered by state-of-the-art CFD) thus provides the initial and boundary conditions for the second. Main CFD inputs to this problem concern the description of the transportation mechanisms to the core: namely, pump start-up or natural circulation after restoration of the water inventory. Relevant parts of the reactor for flow modelling concern at least the downcomer, the lower plenum, and possibly the pipework related to the initial transportation of the slug to the RPV. One-dimensional system codes are not able to simulate these processes realistically; CFD analysis is needed, due to the multi-dimensional, transient nature of the flow, the geometrical complexity of the computational domain, and the requirement of accurately representing the mixing of the different flow streams.

Figure 1a shows the outer surface of a typical CFD model (meshlines removed) for a 3-loop PWR. The mesh



(a) Model layout (b) Boron concentration at core inlet

Fig. 1. CFD Simulation of a Three-loop PWR to Study the Boron Dilution Issue

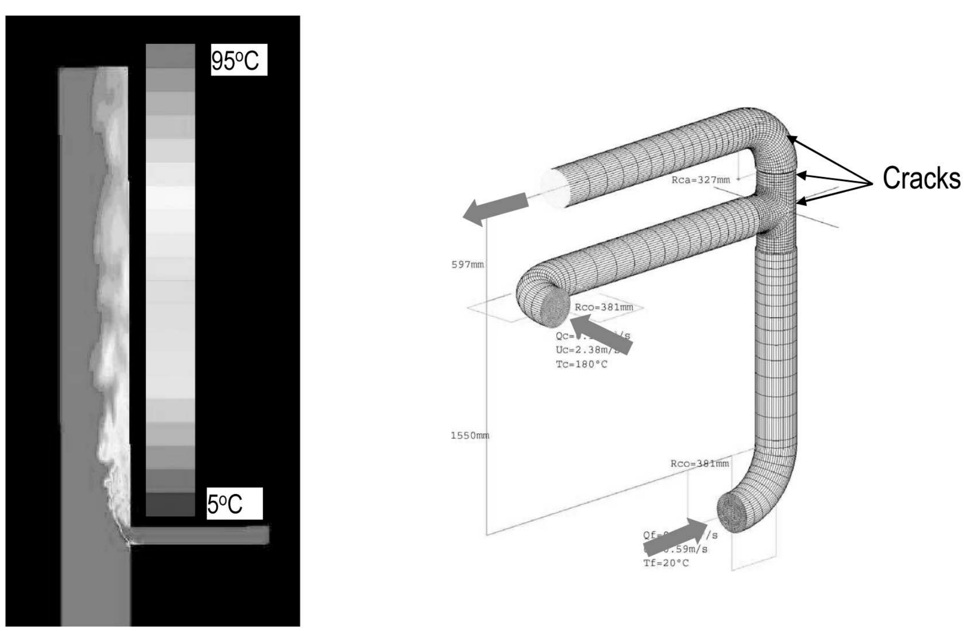
is constructed, in this case, of 6.7 million hexahedral cells. At the start of the transient, a low-boron water slug occupies the region indicated in one of the cold-legs. The flow in all three cold legs is started simultaneously. A profile of the boron concentration at the entrance to the core at the instant the slug arrives is shown in Fig. 1b. As can be seen, there remains a heterogeneous distribution of boron, indicating that incomplete mixing of the cold-leg streams is predicted for this case.

Many CFD validation tests have been performed, based on measured data from a number of experiments: e.g. University of Maryland [19], which formed the basis of the International Benchmark Problem ISP-43, the ROCOM facility at FZD Rossendorf [20], and the Vattenfall 1:5 scale test in Sweden [21]. In addition, boron dilution and general in-vessel mixing have been the subject of the EU-funded programmes EUBORA [22] and FLOWMIX-R [23]. Further details are given in Section 6.1

3.2 Mixing and Thermal Fatigue

Thermal stratification, cycling and striping phenomena may develop in the major components and piping systems of nuclear plants. The phenomena can occur in safety-related lines, such as the pressuriser surge line, the emergency core cooling injection lines, and other lines where hot and cold fluids come into contact with each other. Damage resulting from the associated fluctuating thermal loads to nearby structures has been reported for mixing tees of the feedwater systems, in the reactor clean-up systems, and in residual-heat removal systems. Sometimes, defective valves through which hot (or cold) water leaks into a cold (or hot) water stream are the cause of such fatigue problems. Figure 2a shows a view through the mid-plane taken from a CFD simulation of a scaled mixing tee experiment in which there is a bleed flow from a leaking valve in the branch line. As can be seen, there are coherent, large-scale turbulent motions downstream of the junction. These induce thermal fluctuations in the pipe, a situation that can lead to high-cycle thermal fatigue. The problem is a serious safety concern in respect to ageing and life management of nuclear plants. Coolant temperature oscillations due to turbulent, thermal mixing effects that pose a risk of wall thermal fatigue are reported to be at frequencies up to several Hz [24]. Significantly higher frequencies than these, however, are considered not to pose a risk, as they are strongly attenuated by the thermal inertia of the pipe material.

In general, the common thermal fatigue issues are well understood and can often be controlled or circumvented (by adding internal static mixers, for example). However, some incidents indicate that certain information on the loading in the mixing zone, and its impact on the structural material, is still missing. One such incident occurred at the Civaux-1 PWR in 1998 [25,26], which was worrying in that the plant (1450 MW N4) had only been in operation for 1500 hours, and three similar plants had also been



(a) CFD simulation of a bleed flow event (b) Mesh layout from a study of the Civeau-1 incident

Fig. 2. Using CFD to Study Thermal Fatigue in T-junctions

constructed. Figure 2b shows the mesh layout used to analyse the mixing processes using CFD [27]. The piping arrangement for all four plants was subsequently changed to an earlier design type.

3.3 Hot-Leg Heterogeneities

For the safe running and control of a PWR, it is essential to have, as precisely as possible, knowledge of the real primary flow rates to ensure that they do not exceed the limiting design-basis values. The upper value is derived from mechanical considerations regarding the assembly holding forces and the control rod falling time while the lower value is associated with the DNB (Departure from Nucleate Boiling) risk protection signal. In the plant, the actual primary flow rates are not measured directly but estimated indirectly from internal temperature measurements and overall heat balances.

By far the main source of uncertainty in this procedure (about 10 times greater than from other sources) is related to estimating the average hot-leg temperature. Despite the mixing processes taking place in the upper plenum, important temperature and flow heterogeneities may still be present at the hot-leg instrumentation location, leading to uncertainties in the estimation of the actual average temperature and by inference the actual coolant flow rate. In order to quantify the error, the average temperature of the hot-leg has to be estimated using scale-model tests, from specific plant data, or from CFD calculations.

Direct extrapolation of experimental results to real plant conditions is very difficult [11] and can often result in an overestimation of the uncertainty. The use of such overestimated values in the case of actual plant situations (e.g. core loading) can give results which do not satisfy the specified safety criteria. Advanced methodologies based on CFD calculations can reduce the level of uncertainty. Results to date are encouraging [28]. CFD simulations are able to reproduce qualitatively all the phenomena observed during the experiments: the upper-plenum flow, the temperature contours from the core to the hot legs, and the flow pattern in the hot legs (which is actually composed of two counter-rotating vortices). The main problem impeding further progress is the sheer complexity of the geometry, making the calculations slow and expensive (a situation that will improve with advancements in computer hardware). However, the physical models also need to be improved, and a very fine-scale representation of the turbulent phenomena is required to localise the vortices in the hot-leg. Consequently, application of CFD codes requires validated models to estimate mixing in the upper plenum and vortex development in the hot-leg.

3.4 BWR/ABWR Lower Plenum Flow

There are many pipes in the lower plenum of a BWR or Advanced Boiling Water Reactor (ABWR) reactor (see Fig. 3a). Two phenomena are relevant to NRS. One is the stress induced by flow vibration, which may cause these pipes to fret and perhaps break, and the other is a lack of uniformity of flow between the pipes, which may lead to a non-uniform temperature distribution in the reactor core.

Many internal structures are located close together in the lower plenum. At a time of partial pump operation, which is an accepted mode of operation in an ABWR, inverse flow can occur in the leg attached to the inactive pump. CFD codes are effective in evaluating the flow field in such geometrically complex situations, and significant progress has been made in Japan [29] in support of their ABWR programme (see Fig. 3b). Note that since the study relates to the lower plenum of the reactor, it is a single-phase application.

3.5 Pipe Break

Transient pressure forces occur on structures following a large pipe break and are of importance for various reactor types. Inside the RPV, the decompression waves produce dynamic loadings on the surfaces of the vessel internals, such as the core shroud and core grids of a BWR. This issue is an important example of the need to predict accurately the transient three-dimensional pressure fields in order to estimate the resulting dynamic loads on structures. It is also important to realise that modern structural analysis has to include dynamic loads, even for Loss-of-Coolant-Accidents (LOCAs).

The decompression process is a highly 3-D and transient phenomenon, so it can only be realistically simulated using CFD. During the first phase, before flashing of the water in the RPV begins, a single-phase CFD model can be used, but after flashing has started a two-phase model is necessary to describe the decompression process. From the beginning of the flashing of the water, the two-phase phenomena are dominant.

CFD analysis of a Main Steam Line Break (MSLB) in a BWR was carried out as part of a qualifying programme before the replacement of the core grids at Units 1 and 2 at Forsmark NPP in Sweden [30]. The results indicated a rather complex character of the decompression process; the instantaneous forces were computed to be approximately twice those estimated previously using simpler methods. The results have not yet been validated against experiments, however.

Coupled CFD/FEM analysis has been undertaken in a simulation of one of the HDR experiments, performed at FZK (now KIT), Karlesruhe [31]. Predictions based on a single-phase fluid model, with no possibility of phase change, and with fluid-structure interaction (FSI), compare well with experimental data for the first 100 ms after the break. Thereafter, two-phase phenomena dominate, and to date no simulation has been attempted.

3.6 Hydrogen Distribution in Containments

During the course of a severe accident in a Light Water

Reactor (LWR), large quantities of hydrogen could accumulate in the containment. Detailed knowledge of the containment thermal hydraulics is necessary to ensure the effectiveness of the hydrogen mitigation procedures. Condensation and evaporation on walls, pool surfaces and condensers need to be modelled realistically because the related mass and heat transfer processes strongly influence the subsequent pressure and mixture composition in the containment. In addition, there is the issue of pressure loading to the structures. The mixture composition is very important because it strongly affects the burning mode of the hydrogen and the operation of the PARs (Passive Autocatalytic Recombiners).

Containments have very large volumes and have multiple compartments. A too-coarse nodalisation will not only lose resolution but will smear the temperature, species concentration and velocity fields through numerical diffusion. From a physical point of view, the flow model must also take into account condensation (in the bulk, or on the surfaces of cold walls) with the associated heat transfer to the structures. Unfortunately, condensation models are not yet standardised in CFD codes.

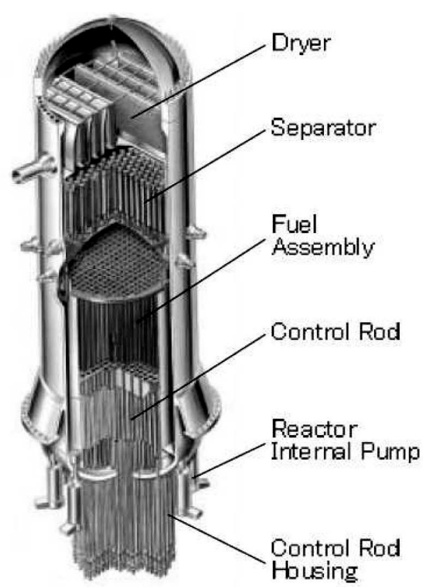
A CSNI State-of-the-Art Report was issued in 1999 [32]. It concluded that current lumped-parameter (i.e. 0-D) models are able to make adequate predictions of the pressure history in the containment and of the average steam content. Predictions of hydrogen distributions were regarded as acceptable but only if safety margins were kept large enough. The benchmark exercise ISP-47 was aimed precisely at validating CFD codes for containment thermal-hydraulics, including the hydrogen risk. Simulations of experiments from the TOSQAN [33], MISTRA [34] and ThAI [35] series were included in the exercise. More recently, tests from the OECD/SETH series [36] have just been released (Dec. 2009); some tests are relevant to the issue of hydrogen distribution in containments and will contribute to this assessment database.

3.7 Chemical Reactions/Combustion/Detonation

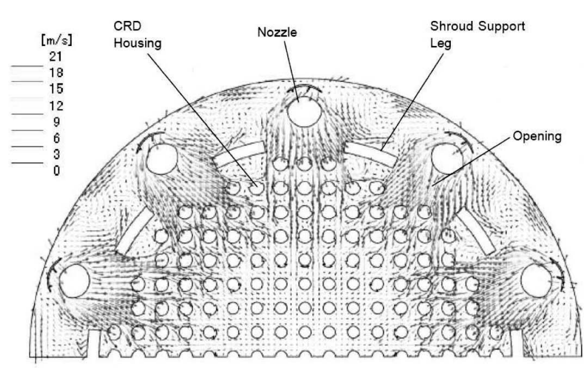
Detonation and combustion in containments may lead to pressure rises that exceed the design specifications. There is also the risk of localised overheating of structures in the case of standing flames. Deflagrations, accelerated flames, or even detonations, can all be envisaged for some accident scenarios.

Deflagration is a very complex phenomenon, involving both chemistry and turbulence. CFD, combined with flame-speed-based deflagration models, can provide insights into the dynamic loadings on the structures. In contrast, detonation processes are relatively simple to model because the very fast front propagation means there is little feed-back from other slower processes, such as chemistry, fluid flow and structural deformation. However, simulation of shock-wave propagation should also account for multiple reflections and superposition of the waves. In principle, CFD has the capability to follow these phenomena.

The CFD code FLUENT [37] has been used to calculate



(a) ABWR general layout



(b) Flow field in lower plenum

Fig. 3. ABWR (Flow Field Courtesy of Shiina et al., 2003)

the hydrogen distribution in a BWR containment, in combination with DET3D [38] for the 3D detonation simulation, and with ABAQUS [39] for the structural analysis and load evaluation. There have been many applications of compressible CFD solvers to model detonations in large-scale geometries: for example, the RUT experiments from the Kurchatov Institute [40], also some calculations of fast deflagrations in a simplified EPR (European Pressurised Water Reactor) containment were performed in the framework of the 5th FWP Project HYCOM [41]. Hydrogen deflagration models and CFD codes were also evaluated in the 4th EU FWP programme HDC [42].

3.8 Aerosol Deposition in Containments

Following a severe reactor accident, fission products would be released into the containment in the form of aerosols. If there were a subsequent leak in the containment barrier, these aerosols would be released into the environment and pose a health hazard. The most conservative assumption is that all the fission-product aerosols would eventually reach the environment. A more realistic assessment can be made by studying the detailed processes which govern the initial core degradation, fission product release, aerosol-borne transport and retention in the coolant circuitry, and aerosol dynamics and chemical behaviour in the containment. CFD can be used to model these processes by employing Lagrangian tracking of the aerosols.

Some CFD calculations have been performed in simulations of tests from the PHEBUS-FP facility at CEA Cadarache [43]. The facility provided prototypic reactor conditions from which integral data on core degradation, fission product release, aerosol-borne transport and retention in the coolant circuit, and aerosol dynamics and chemical behaviour in the containment. However, all data from the tests are of integral type, sufficient only to validate lumped-parameter codes. There remains a distinct lack of data suitable for CFD validation in realistic geometries (see also Section 7.3).

3.9 Atmospheric Dispersion

Again, following a severe reactor accident, radioactive release to the atmosphere could ultimately occur, which may represent a health hazard for the installation workers and the surrounding population. Atmospheric release of nuclear materials (in the form of both aerosols and gases) implies air contamination, at first on-site and later off-site. The atmospheric dispersion of such material in complex situations, such as buildings being in close proximity to the reactor, is a difficult problem to analyse, but one that is important for the safety of the people living and working in such areas. Dispersion models, which are used to estimate the levels of radio-activity, require meteorological data as input. Typical examples of such data are atmospheric velocity fields and temperature distributions.

Atmospheric motion and dispersion are 3-D in character, turbulent and unsteady; CFD is the traditional approach in the investigation of such flows. On-site simulations must take account of the proximity of nearby buildings and effects due to the wind and weather (i.e. radiation heating by the sun, precipitation, etc.). Off-site, account has to be taken of the topography of the landscape, night-and-day effects, stratified layers, etc. However, within the realm of the physical phenomena, the major challenge lies in turbulence modelling. The flows are highly 3-D, unsteady and are accompanied by strong streamline curvature, separation, and the generation of vortices. The redeeming feature is that atmospheric dispersion is not unique to releases from nuclear power plants, and much progress has been made in other disciplines, in particular in the release of toxic chemicals from non-nuclear industrial plants. As a consequence, little attention has been paid to the issue by the nuclear community except in integral form. The use of best-estimate methodology, such as CFD, would improve the reality of the predictions should more localised information be required.

3.10 Flow-Induced Vibrations

Flow-induced vibrations in steam generators (SGs) have been studied for many years, since tube rupture could have serious consequences due to loss of coolant and because of the risk of direct release of radioactive material to the environment. Tube vibration caused by dynamic forces (as generated in the U-bends of standard PWR SGs) may initiate mechanical damage due to fretting, wear, and fatigue. Similar concerns are being expressed in the context of the radial reflectors of Advanced Pressurised Water Reactors (APWRs), since excessive vibration could result in rupture of the fuel-pin cladding, the first barrier against release of radio-active material.

If the core barrel of the APWR is set into vibration by the turbulent flows in the downcomer, the vibrations would be transmitted to the radial reflector through the water filling the space between them (Fig. 4). If the radial reflector vibrates, the grid supporting the outermost fuel bundles may make contact with it, and when the grid vibrates, the fuel cladding could wear through fretting.

In order to estimate the level of vibration of the radial reflector with sufficient accuracy, it is necessary to calculate the pressure fluctuations generated by the turbulent fluctuations in the downcomer correctly, since these are the driving forces for the vibration. The following two methods are available for using CFD to evaluate the vibration between fluid and structure:

- The vibration between the fluid and the structure is calculated directly by the coupled use of a CFD code and a structural analysis code, perhaps with the fluid-structure interaction described by a Lagrangian moving-mesh technique;
- The vibration between the fluid and the structure is

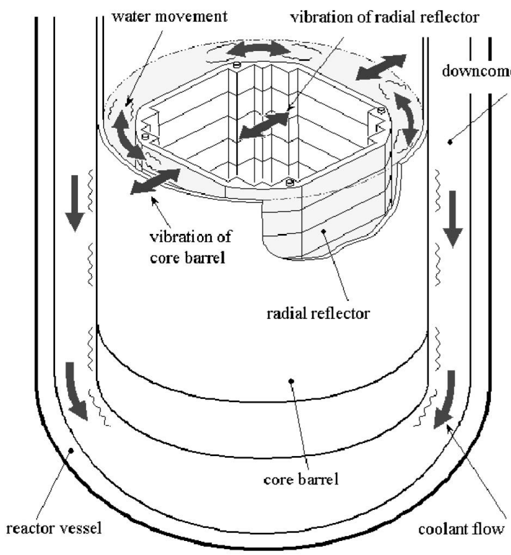


Fig. 4. Schematic Illustrating Flow-induced Vibration for the Radial Reflector in an APWR

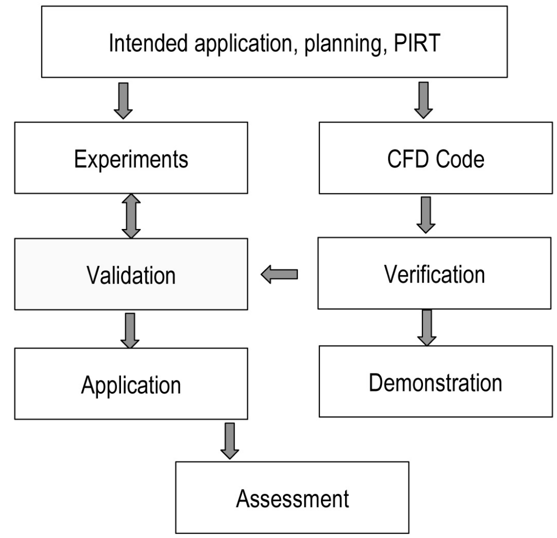


Fig. 5. Block Diagram of a Numerical Assessment Procedure

calculated by the structural analysis code directly, modelling the water between the core barrel and the radial reflector simply as an additional mass, and imposing the downcomer pressure fluctuations calculated by the CFD code as load conditions.

The latter method is by far the more practical, but the emergence of corporate links between the structural dynamics FEM (Finite Element Method) code ANSYS [44] and the established industrial (Finite Volume) CFD codes FLUENT [37] and CFX [45] brings the prospect of fully coupled simulations for addressing such problems significantly closer. In addition, the FEM code ABAQUS has just released details of a fully coupled CFD module [46].

4. VERIFICATION, VALIDATION AND ASSESSMENT

Having identified the potential application areas, it is necessary to define the steps needed to produce reliable CFD predictions. Modern CFD codes consist of hundreds of thousands of lines of coding, written by different programmers. It is inconceivable there are no ‘bugs’ in the program. In addition, access to the code’s stores is often made via a Graphical User Interface (GUI). The transfer of information from the GUI to the central solver, and vice versa, must be absolutely accurate. Finally, the code’s documentation must faithfully represent what is actually coded. Paraphrasing Oberkampf and Trucano [47] where necessary, the process of correcting all these faults is called *verification*. Formally, verification is defined as “the process of determining that the implementation of a physical model or numerical method accurately represents the developer’s conceptual description of the model and the solution to the model”. Basically, this means that the coded equations are being solved correctly, there being no requirement at this stage to demonstrate whether or not the equations represent ‘physical reality’; this issue is taken up under the heading *validation*.

Verification is principally the responsibility of the code developers, though users can participate by performing verification calculations. This process consists of examining the model implementation through comparison of code predictions against exact analytical results, manufactured solutions [48], or previously verified higher accuracy simulations. Ideally, testing of all relevant implementation aspects of the CFD code should be undertaken as confirmation that accurate and reliable results can be obtained from the mathematical models programmed into the code.

But how good are the mathematical models in regard to representing physical reality? It is the task of the *validation* procedures to address this key question, one that can only really be successfully answered by comparing code predictions against measured data. The message is put very succinctly by Roache [49] as: “verification deals with mathematics, validation deals with physics”.

The WG2 Writing Group defined the word *assessment* as “the expression of belief (based on validated calculations) that a given computer code is able (when properly used) to simulate with acceptable fidelity a given set of situations (at least parts of a nuclear reactor transient)”. Assessment therefore requires validation of an already verified computer code against suitable experimental data. Figure 5 graphically depicts the necessary stages to be followed for a successful assessment. The procedure starts with a

Phenomena Identification and Ranking Table, or PIRT [50]. The PIRT approach originated as part of the US NRC's methodology for the use of best-estimate simulation codes in the licensing of nuclear power plants. Phenomena and processes are ranked in the PIRT based on their influence on primary safety criteria, and subsequent efforts are focused on the most important of these. This process has broadened over the years, and is now also used outside the nuclear community as an important component of any validation procedure.

Step 1. *Careful definition of the objectives*. It is often more effective to define a series of specific PIRT exercises (e.g. boron dilution) rather than trying to cover everything with a very general PIRT exercise (e.g. SB-LOCA).

Step 2. *Appointment of a panel of experts*. Both technical and managerial expertise should be represented on the panel. At least one member should have a primary focus in each of the following areas, relevant to the scenario and system under study:

- Experimental programs and facilities;
- Simulation code development (numerical implementation of physical models);
- Application of relevant simulation codes to the present and similar scenarios;
- Configuration and operation of the system under study.

Step 3. *Review of objectives, system and scenario*. With this done, a list of parameters of interest can be compiled.

Step 4. *Identification of relevant existing information*. This should primarily be experimental data, and results of related analyses. The process relies heavily on the knowledge and experience of panel members but can be broadened, if required.

Step 5. *Identification of phenomena and processes associated with the system under the specified scenario*. This step is self-explanatory.

Step 6. *Ranking of the phenomena*. This is the end-point, but is also the most important aspect of the procedure. The ranking can be done in terms of a L/M/H priority assignment but with subdivisions if necessary. The process may need to be iterative if the situation demands it.

There are circumstances in which no validation calculations of the situation with a given computer code have been undertaken so far or that experimental data are sparse or non-existent. In such cases, the CFD code can be *demonstrated* to have the capability of simulating the situation, but a route to a final safety assessment is not possible, since the adequacy of the physical models in the code to represent the relevant physical phenomena has not been scrutinised; that is, the validation step has been bypassed. The procedure is illustrated graphically in Fig. 5, which emphasises the point that there is no continuous link between the verification and assessment boxes without performing the validation step. All that can be done in this case is to *demonstrate* a capability to perform the allotted task.

Any assessment matrix should be strictly problem-dependent: that is, any particular matrix must contain at least part of a computational path (numerical algorithm and/or physical model) considered appropriate for the intended application of the code. Ideally, a separate assessment matrix should be prepared for every selected nuclear safety issue where CFD simulation is deemed to be beneficial. This is a very demanding task. Fortunately, there will be many points of overlap between such groups of matrices, since the same numerical algorithm and physical models will often be used in different applications. It is worthwhile therefore to look at the common ground between nuclear and non-nuclear validation material.

5. EXISTING ASSESSMENT BASES (NON-NUCLEAR)

Major sources of information identified by the WG2 Writing Group are elaborated below under appropriate section headings. Some of the websites referenced allow free access to data for code validation; they sometimes propose CFD reference calculations, and they sometimes ask people to participate to the enhancement of the database by submitting their own cases. In this way, the CFD community has ready access to an ever increasing body of information to act as an assessment base for their activities. At present, the activities are orientated primarily towards the aerospace and aerodynamics communities, but they at least demonstrate the seriousness of the commitment to *quality and trust* in CFD. It was part of the mandate for WG2 to expand the concept to serve the nuclear community too (see Section 8). Several available general-purpose databases comprising experimental data catalogued by the group are listed here.

5.1 Validation Tests Performed by Major CFD Code Vendors

The code vendors identified are those who promote general-purpose CFD, but who also have customers in the nuclear industry: namely, ANSYS-CFX [45], STAR-CCM+ [51], FLUENT [37] and (to a lesser extent these days) PHOENICS [52]. Also, included are codes written specifically for nuclear applications, though not always available for general use. The principal ones are TRIO-U [53] and SATURNE/NEPTUNE-CFD [54]. Other CFD software with specialisations in certain areas, but with no established nuclear base, such as OpenFOAM [55] and ACE-CFD+ [56], were excluded from the list though they may be added later, as appropriate.

Each of the four main industrial CFD vendors operates in a highly competitive commercial environment, and each is acutely aware of the state of development of their major competitors. Consequently, such a sensitive item

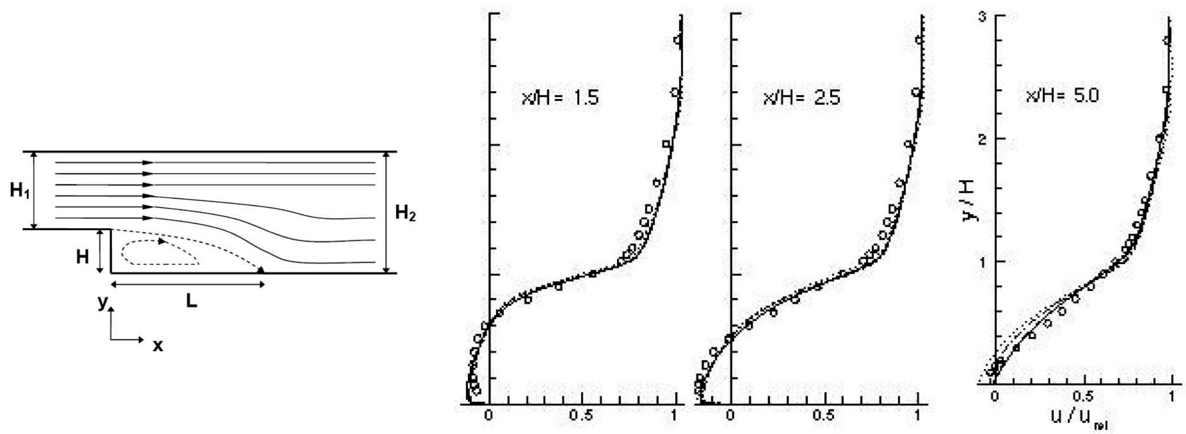


Fig. 6. Backward-Facing Step Benchmark on the ERCOFTAC Database: Configuration and Velocity Profiles (Re-attachment at x/H = 5)

as *validation*, which might lead them into unwelcome code-to-code comparison exercises, was initially treated rather sceptically. In addition, a validation activity may have been performed at the request of a particular customer, and the results may have been restricted or may not (yet) have been published. More recently, as their customer base became more aware of the common goal of *quality and trust in CFD*, the companies have become more open and have often actively participated in international benchmarking activities. The best source of information on specific validation databases is through the respective websites. Here, one finds documentation, access to the workshops organised by each company, and to the conferences and journals where customers and/or staff have published validation material. The list of validation cases is clear evidence that commercial CFD is a well-founded technology. It should also be noted that even codes explicitly written for the nuclear community also include basic (often academic) validation cases, just like those codes from the general industrial (commercial) area.

5.2 ERCOFTAC

The European Research Community on Flow, Turbulence and Combustion (ERCOFTAC) is an association of research, educational, and industrial groups whose main objectives are to promote joint efforts through centres and industrial application of research, and to create Special Interest Groups (SIGs) in certain areas [57]. One such special interest group is the ERCOFTAC Database Interest Group (Dbig). The database was started in 1995 and is actively maintained by the University of Manchester, UK. It contains experimental as well as high-quality numerical data relevant to both academic and applied CFD applications. ERCOFTAC holds regular workshops on refined turbulence modelling around Europe, information from which is used to update and refine the database. The Classic Data Base is open to the public (though a simple registration procedure has to be followed before data may be downloaded). There are more than 80 documented cases, containing either experimental data or with highly accurate DNS (Direct Numerical Simulation) data available. Each case contains at least a brief description, some data to download, and references to published work. Some of the cases could be used also in NRS applications, such as flow in curved channels, mixing layers, separated flows, impinging jets, and flows through tube bundles.

Cases have been categorised by flow type, for convenience:

Free Turbulent Flows:	homogeneous flows; free shear flows; interacting shear flows;
Flows Around Bodies:	two-dimensional and three-dimensional configurations;
Semi-Confined Flows:	2-D boundary layers; 3-D boundary layers; wall jets; flows around bodies interacting with boundaries; free-surface flows;
Confined Flows:	flows with/without separation; cavity flows; unsteady flows.

As illustration, a classic example is flow over a backward-facing step (Fig. 6), which examines several important aspects of turbulent flows: separation of a turbulent boundary layer, re-attachment of the boundary layer, recirculation, and the occurrence of secondary separation regions. Many such flow situations also occur in the nuclear thermal-hydraulics area. There is a wealth of experimental data for increasing Reynolds number, and simulations include Direct Numerical Simulation (DNS), Large Eddy Simulation (LES), and different Reynolds-Averaged Navier-Stokes (RANS) models. Care was taken that upstream conditions were fully developed, and the velocity profile measured, since such information is needed to specify the input boundary conditions for the actual CFD simulation. Velocity profiles downstream of the step were also measured, capturing the recirculation region and beyond. Typical measured-versus-calculated data are shown in Fig. 6.

5.3 QNET-CFD KB

QNET-CFD KB developed from the QNET-CFD web-based thematic network, which was a part-funded European project to promote the quality of CFD and trust in the industrial application of CFD [58]. Several years were spent in assembling and collating knowledge and know-how across a range of hierarchically structured application areas: aerodynamics, combustion and heat transfer, chemical and process engineering, thermal hydraulics and nuclear safety, civil construction and HVAC (heating, ventilation and air conditioning), and environmental flows and turbomachinery. Specific NRS items include: buoyancy-opposed wall jet, induced flow in a T-junction, buoyant gas-air mixing, mixed convection in a reactor (containment gas mixing), spray evaporation in turbulent flow, combining/dividing flow in a Y-junction, and downward flow in a heated annulus. For each *Application Challenge*, its description, test data, CFD simulations, evaluation, best practice advice, and information on related underlying flow regimes are all available.

Between 2000 and 2004, a Knowledge Base containing 43 Application Challenges was established, later expanded, and finally brought online by means of a Wiki-based website, which had been developed from the prototype pioneered by the QNET-CFD network. The Wiki pages now come under the administration of the ERCOFTAC organisation [57].

5.4 NPARC Alliance Data Base

Chiefly orientated towards the aerodynamics community, the CFD Verification & Validation section provides a tutorial [59] as well as measurements and data for CFD cases. There is a link to the data archive of NASA, which is particularly useful. High quality data are available in the following areas: incompressible, turbulent flow over a flat plate, RAE 2822 transonic airfoil, S-Duct, subsonic conical diffuser, 2D diffuser; supersonic axisymmetric jet flow, incompressible backward-facing step, ejector nozzle, transonic diffuser, hydrogen-air combustion in a channel, two-stream mixing, and laminar flow over a circular cylinder. Many of the basic flow configurations are relevant to NRS analyses at a fundamental level.

5.5 AIAA (American Institute of Aeronautics and Astronautics)

The society participates in the definition of standards for CFD in its *Verification and Validation Guide* and has important links to websites containing lists of references (papers, books, author coordinates) related to CFD verification and validation. Also, there are various links to other websites, containing information of (principally) aeronautical interest. Some of these links may be useful for CFD validation, but would need sifting for relevance to NRS.

In summary, extensive validation data for CFD simulations are available in terms of basic generic flow configurations. The data form the building blocks of the models that ultimately find their way into both the general-purpose and nuclear-specific CFD simulation software currently being used to perform NRS simulations. Because the fluid flow and heat transfer situations encountered in NRS studies are, for the most part, mirrored in other industrial applications, the nuclear community can benefit from the quality and trust in CFD established in such non-nuclear areas. Of course, situation-specific data are also needed for nuclear safety analyses, and such data bases were also catalogued by the WG2 group. An overview is given in the next section.

6. EXISTING ASSESSMENT BASES (NUCLEAR)

6.1 Boron Dilution

Experiments focussing on the boron dilution event (described is Section 3.1) generally try to reproduce the mixing in the reactor downcomer and lower plenum upstream of the reactor core inlets. The databases are well-established and have been used previously for benchmarking exercises, as Table 2 illustrates.

Under the terms of an OECD benchmark exercise, International Standard Problem ISP-43, two sets of experiments performed at the University of Maryland facility UM2x4 Loop were made available for numerical analysis. Originally, these were for “blind” analyses, meaning that the test data were not available for comparison at the time the numerical simulations were carried out, but several post-test simulations have also been published.

Table 2. Boron Dilution Database

Test	Reactor Type	Scale	International Benchmark Activity
University of Maryland (US)	Babcock and Wilcox PWR	1/5	ISP-47
ROCOM (Germany)	Konvoi PWR	1/5	FLOWMIX-R
OKB Gidropress (Russia)	VVER-1000	1/5	FLOWMIX-R
Vattenfall (Sweden)	Westinghouse 3-Loop PWR	1/5	FLOWMIX-R

The UM2x4 Loop is a scaled-down model of the Three Mile Island Unit 2, Babcock & Wilcox PWR. Sixteen redundant Test A (front mixing test, with an infinite slug of cold water entering the RPV) and six redundant Test B (slug mixing test, with a finite-volume slug of cold water entering the RPV) experiments were performed. Detailed boundary conditions were provided for the analysts, and time histories of temperatures at nearly 300 positions at 11 elevations within the downcomer and lower plenum were ultimately made available to the participants. The model of the RPV, with positions of thermocouples marked, is shown in Fig. 7a. Ten participants from eight countries submitted numerical results to the blind-calculation phase of the benchmark. The CFD codes featured were CFX-4, CFX-TASCflow, FLUENT and TRIO-U.

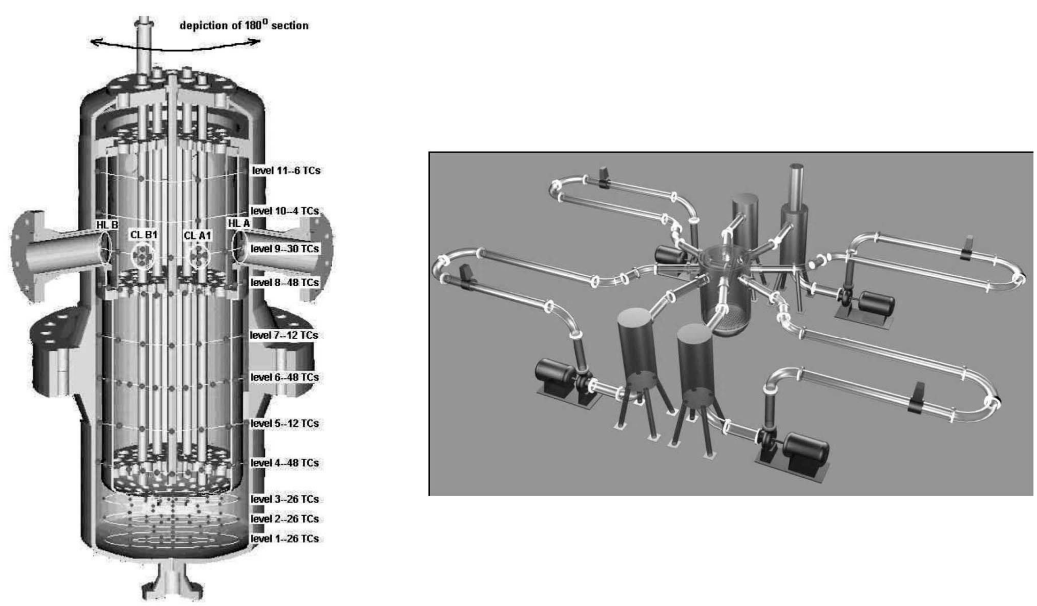
The ROCOM [60] facility (Fig. 7b) consists of four loops, with fully controllable coolant pumps. In contrast to the Maryland tests, demineralised water was used in these tests, supplemented by the injection of slugs of a tracer solution (diluted salt) into one loop. The salt concentration was measured by means of wire-mesh sensors [61]. Of these, one was installed in the cold-leg inlet nozzle of the disturbed loop (232 measuring points), two were placed in the downcomer just below the inlet nozzles and before the entrance to the lower plenum (2x232 measuring points), while the fourth sensor was integrated into the lower core support plate, with one measuring position at each fuel element location. Further, all four outlet nozzles were also equipped with sensors (4x232 measuring points). Laser Doppler Anemometry (LDA) was applied for the velocity measurements. Data from selected tests were made available for CFD analysis within the EU Framework Programme FLOWMIX-R [23]. Further test data from scaled facilities were also made available to participants in the FLOWMIX-R project. These originated from the OKB Gidropress facility in Russia and the Vattenfall experiment in Sweden.

All the boron-dilution test facilities model the corresponding original reactor at a scale of 1:5. Of the various CFD analyses carried out, the blind calculations performed in the context of the ISP-43 produced large discrepancies between numerical and measured data, even for participants using the same CFD tool [19]. Consequently, the question of whether CFD is capable of being used reliably for this particular NRS problem is not yet resolved.

6.2 Pressurised Thermal Shock (PTS)

During a Small-Break Loss of Cooling Accident (SB-LOCA) scenario in a PWR, Emergency Core Cooling (ECC) water is injected into the cold-leg pipe and mixes with any water remaining in the pipe. The combined streams flow towards the downcomer, where further mixing takes place. In the case of incomplete mixing of the streams, the cold water from the ECC line will come into direct contact with the RPV wall and may lead to large temperature gradients inside the vessel material, generating high thermal stresses. Knowledge of such thermal loads is important for plant-life extension assessment, since during its service life the RPV will have become subject to radiation-embrittlement.

Most attention has been paid to the two-phase PTS



(a) Maryland (ISP-43 Benchmark) (b) ROCOM (FLOWMIX-R Project)

Fig. 7. Two Boron Dilution Experiments for Which Measured Data were Released

event (Fig. 8), with high pressure injection from the top into a partially filled cold-leg pipe (a scenario of relevance to French PWR designs) though there remain thermal shock issues associated with the single-phase event too, in which either the pipe is full, or the injection is below the water surface (as in the German Konvoi and Russian VVER designs). An extensive experimental database for single-phase fluid mixing relevant to the PTS issue was compiled by Theofanous and Yan [62]. The information is summarised in Table 3, building on data supplied by Wolf [63]. Since this time, the major PTS test facilities have concentrated on the two-phase PTS issue. According to information compiled during the ECORA project [64], experimental data for validation of a CFD code should be complete in regard to geometry, boundary and initial conditions, well analysed with respect to the physical phenomena involved, be of high quality (i.e. accurate within specified error bounds, repeatable and consistent), and, most importantly, be publicly available. The WG2 Writing Group has not yet undertaken scrutiny of the existing PTS database in terms of these criteria.

Though yet to be performed, the TOPFLOW-PTS [65] experiments being undertaken at FZD, Rossendorf in Germany are worth mentioning in advance. Currently, air/water experiments are being conducted in preparation for the steam/water tests to be carried out in the near future. The facility has a unique feature in comparison to other test facilities for which PTS studies have been performed (e.g. FORTUM, ROCOM). The scale is 1:2.5, with the test section located inside a pressure vessel of length 7m and 2.5m inner diameter (Fig. 9). Experiments can be carried out at up to 5 MPa pressure, but parts of the test section can be constructed of glass due to pressure equalisation, thus enabling full visualisation access. The facility is highly instrumented with thermocouples, heat-flux probes, wire-mesh sensors, local void-fraction probes, high-speed cameras, infrared cameras, and local conductivity probes. The geometry for the first tests to be undertaken is based on the French CPY 900 MWe reference plant. It is planned to operate the test mock-up in steady-state conditions, with and without mass transfer due to condensation, as well as in transient operational mode. Access to the data from the tests will be restricted initially to the partners in the consortium who have financed the series, though some will be released on a broader platform in the context of the EU 7th Framework Program NURISP [66]. A wider distribution of data may become possible in due course.

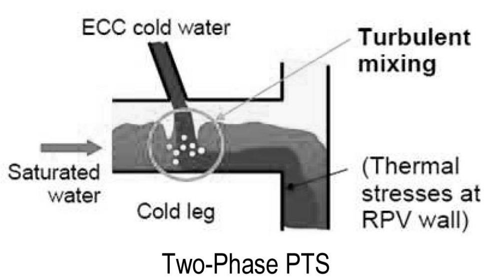


Fig. 8. Schematic of the Two-phase PTS Event

6.3 Thermal Fatigue

Flow-induced failures of parts of structural components of NPPs caused by high-cycle thermal fatigue include Genkai Unit 1 (Japan), Tihange Unit 1 (Belgium), Farley Unit 2 (USA), PFR (UK), Tsuruga Unit 2 (Japan), and Loviisa (Finland). As a result of these incidents, considerable research effort has been devoted to the phenomenon, and both experimental and numerical information has been gathered to aid understanding. Thermal fatigue (or thermal striping) is studied mainly for two geometric configurations: T-junctions and for two or more parallel jets in contact with a neighbouring structure. Under this latter category is included the thermal striping threat to the RPV caused by PTS (See Section 6.2). For both types, the problem is complex, involving

Table 3. Pressurised Thermal Shock Database

	Creare 1:5	Purdue 1:2	Creare 1:2	IVO 2:5	HDR 1:2, 1:4
Scaling	Froude 1:5	Froude 1:2	Froude; 1:2	Froude; 1:2.56	Froude; 1:2, 1:4
Downcomer geometry	planar	planar	planar	semi-annular	annular, complete RPV
Downcomer gap (mm)	46	127	137.2	61c	150
Downcomer width (mm)	670	1180	1616	1840	
HPI-nozzle (mm)	51 top	108 top	20.9 top	27 bottom	50 2 nozzles top 1 nozzle side
No of cold legs	1	1	1	3	1

several scientific disciplines (and consequently several types of computer codes): calculation of the velocity and temperature fields in the fluid, the temperature fields in the solid materials, estimation of the associated mechanical stresses, and the behaviour of cracks in the solid. Any experimental database should reflect and comprehensively cover all of these disciplines. Moreover, coupling between the temperature fields is two-way, which means fluid-dynamic and structure-dynamic computations have to be carried out simultaneously, the data from each being appropriately interfaced.

For T-junctions, the EU 5th FWP project THERFAT [67] provided transient, measured data from experiments conducted by the German company Siempelkamp SPG. The tests covered visualisation measurements in glass models, electrical conductivity measurements in glass models using salt water to represent density differences, and localised near-wall temperature measurements in steel models. Similar experiments have been carried out elsewhere. At the Paul Scherrer Institute (PSI), test data are available from experiments performed in glass tubes using ionised and de-ionised water to identify the main and branch streams in the mixing zone downstream of the junction and conductivity measurements using wire-mesh sensors to measure the degree and character of the mixing taking place [68]. The experimental set-up is shown in Fig. 10a. In this configuration, both pipe branches are oriented horizontally. Contours of the normalised conductivity differences in the plane of the wire-mesh sensor, compiled from data from the 216 measuring positions of the sensor, is shown in Fig. 10b, clearly depicting the mixing zone between the two streams.

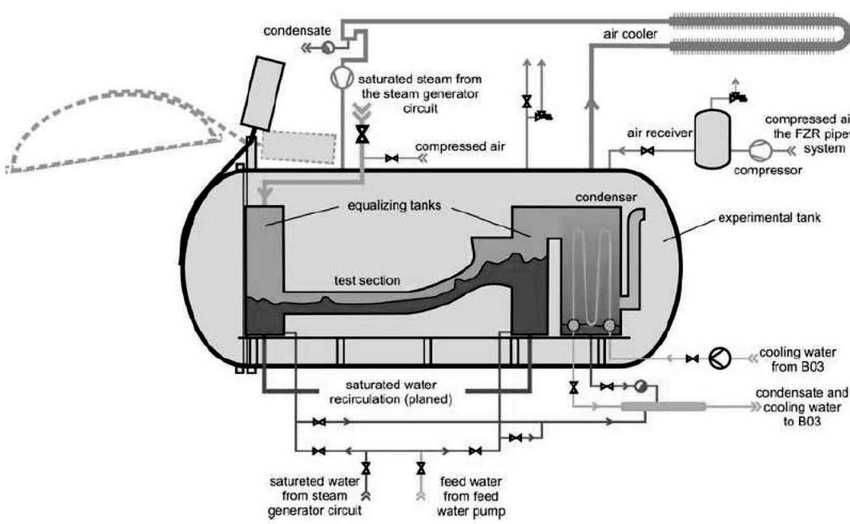
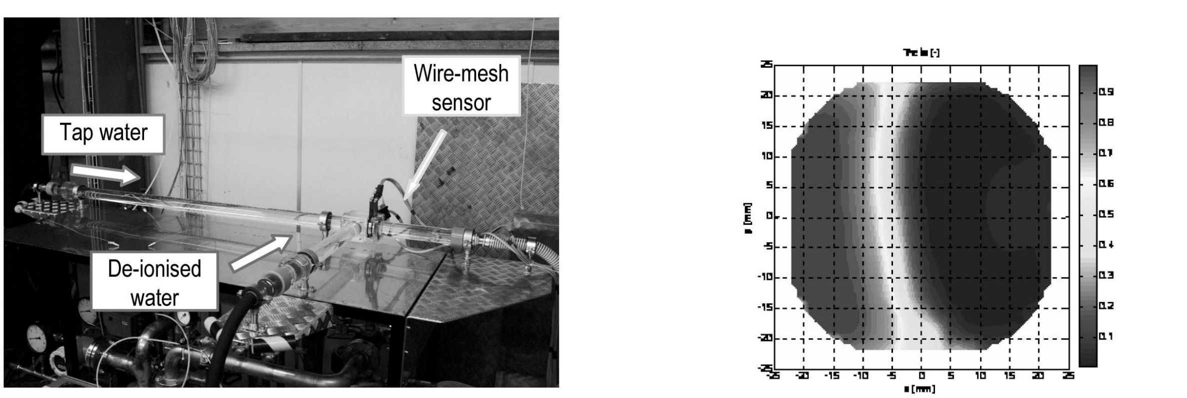


Fig. 9. TOPFLOW-PTS Experimental Layout (FZD, Rossendorf)



(a) Experimental set-up

(b) Conductivity plot derived from wire-mesh sensor readings

Fig. 10. PSI T-junction Experiment

Very careful T-junction tests have been performed at the Vattenfall Älkarleby Laboratory in Sweden. The test section is made of Plexiglas, the main pipe being horizontal and the branch pipe vertical (Fig. 11). The tests were performed with a temperature difference between the inlets of about 15K. Temperatures near pipe walls were measured using thermocouples, and velocity profiles in both inlet pipes and downstream of the junction were measured using Laser Doppler Anemometry (LDA). Data from a test performed in November 2008 have been used to launch a major blind benchmarking activity under the sponsorship of the OECD Nuclear Energy Agency [69]. Participants who supplied CFD simulation results to the organisers before a specified deadline (April 30, 2010) were given access to the measured data (see also Section 8.2).

7. GAPS IN TECHNOLOGY AND ASSESSMENT DATABASES

An assessment matrix for a given application should comprise three groups of items: (i) a verification programme in which CFD predictions are compared against analytical, manufactured, or highly-accurate solutions, (ii) validation experiments and accompanying CFD simulations, and (iii) demonstration simulations, possibly together with data from mock-up experiments. More than 20 NRS-specific cases, which the WG2 Writing Group considered good candidates to substantially benefit from CFD, are listed in Table 1. In the context of these, a number of gaps in the knowledge base were also identified. Some topical examples are described here.

7.1 Isolating the CFD Problem

Traditional 1-D system codes need to be "manipulated" to take into account 3-D effects, when this aspect needs to be taken into account during a particular safety analysis. For example, flows in the upper and lower plena and downcomer of the RPV, and to some extent the core region, are all 3-D in character, particularly if driven by a non-symmetric loop operation. Natural circulation and mixing in compartments of a containment volume are also 3-D phenomena. In all such cases, it is expected that detailed 3-D CFD computations would produce more trustworthy results than are possible using traditional 1-D system codes. However, there are often strong feedback effects from the system parameters, and it is presently inconceivable that CFD will be able to be applied to the entire system. Rather, a "stand-alone" CFD calculation, performed as part of a broader system simulation is all that can be attempted. However, this approach requires specification of the initial conditions for the velocity and temperature fields for the CFD part of the simulation.

The most cost-effective way of doing this is to use the system code to provide input data to the CFD simulation in terms of (transient) inlet boundary conditions, and then run the CFD program in isolation. However, the problem remains of specifying the initial conditions (of velocities and field variables) for the CFD computation within the 3-D domain. To complete the link, the procedure has to be extended by feeding averaged exit boundary conditions from the CFD computation to the system code, and then the system analysis has to be continued. This means interfacing a CFD module to an existing system code in order to perform a localised 3-D computation within the framework of an overall 1-D description of the circuit.

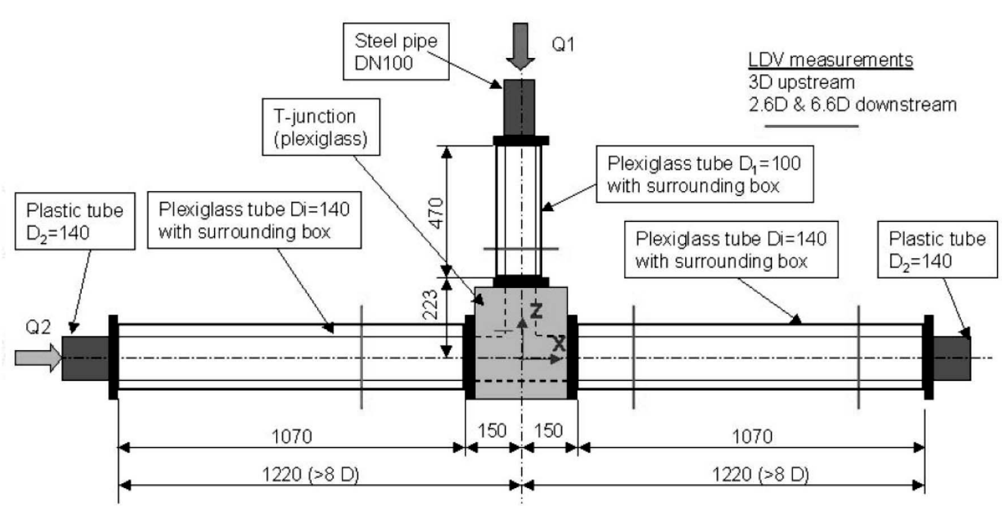


Fig. 11. Set-up of the Vattenfall T-junction Experiment

The best way forward appears to be to directly couple a CFD code to the system code. A summary of current efforts in this direction is described in the next sub-section.

7.2 Coupling CFD with System Codes

Accepting the fact that performing complete nuclear reactor simulations are beyond the capabilities of present hardware devices if a CFD code is used alone, use of a less detailed and less demanding system analysis code to produce boundary conditions for the CFD code is now widely accepted as the only practical alternative. Consequently, links have been established between major system and CFD (or CFD-like) codes. Examples are: RELAP5 to COBRA/TF [70], ATHLET to FLUBOX [71], ATHLET to CFX-4 [72], RELAP5 to CFX-4 [73], RELAP5 to FLUENT [74], and TRACE to CFX-11 [75]. The coupling may be performed via an Executive Program, which monitors the progress in each code, determines when the codes have converged, governs the information interchanges between the codes, and issues instructions to allow each code to progress to the next time step. An alternative is to allocate a master/slave status to the two codes, and control the data exchanges via the master program.

A first validation matrix has been set up for the RELAP5-3D/FLUENT coupled code (which was originally intended for application to pebble-bed modular reactors and other high-temperature gas reactor systems). The matrix involves the simulation of basic flows, such as turbulent flow in a pipe section, flow over a backward-facing step with heat transfer, flow through a pebble-bed core (porous medium approach), and neutronic-fluid interaction within the core. Generally, good progress is being made in this area, though it should be recognised that it is not sufficient to validate the system and CFD codes separately: the coupled code also has to be validated, and the validation process may have to involve integral-type data from system-code benchmark exercises.

7.3 Aerosol Transport in Containments

In a recent PIRT-type exercise [17], aerosol deposition in containments was ranked ahead of thermal fatigue in terms of generic interest, but ironically there are virtually no data from the nuclear area useful for CFD validation. Possible experimental databases could include OECD/NEA activities in the field of aerosol behaviour, such as ISP-37 (VANAM M3 Aerosol Behavior in the Battelle Model Containment [76]), the AHMED Code Comparison Exercise [77], and ISP-44 (KAEVER test facility, VTT, Finland [78]). However, the most cited reference remains the Phebus FP Severe Accident Experimental Program at CEA Cadarache [43], which reproduces (at scale) a core meltdown accident in a French-design 900 MW PWR. Aerosols were released under severe-accident conditions into a mock-up containment. Figure 12 shows a schematic of the facility. Though CFD codes were used within the PHEBEN2 EU-supported project based on the PHEBUS FPT0 and FPT1 experiments, no local measurements of aerosol deposition are available against which to validate the CFD aerosol deposition models.

Results from the PHEBUS tests indicate that the coupling between the thermal-hydraulics and the aerosol physics was rather weak; whereas, in a real plant, where there is more opportunity for stratification, the coupling could play a stronger role in determining local aerosol concentrations. The CFD codes CFX 4.3, CFX 5.7 (FPT1

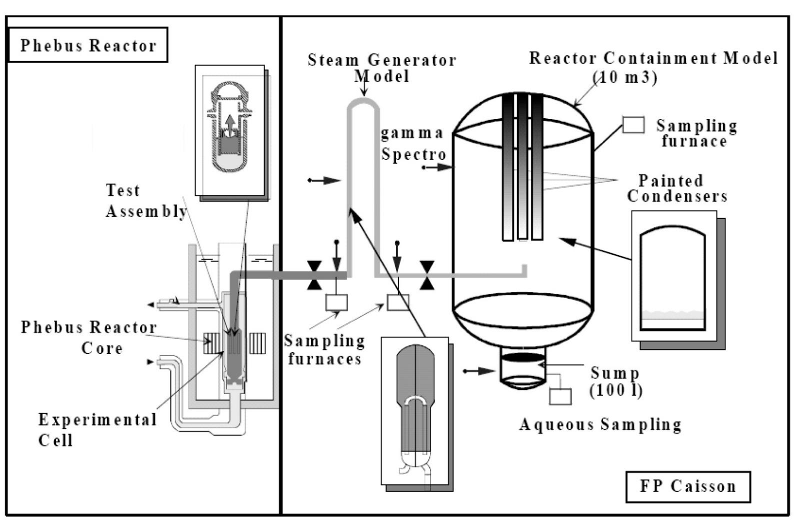


Fig. 12. Schematic of the Phebus Test Facility

only) and TRIO VF have been used for the analysis, but there were problems with the comparison of measured values against numerical predictions, since only a few internal temperature measurements and no velocity measurements were available from the PHEBUS tests. Overall, the case for CFD playing an essential analysis role appears not to be proven, which may explain the clear lack of drive towards producing quality validation data for CFD models in this context.

7.4 Stratification and Buoyancy Effects

Buoyancy forces develop due to heterogeneous density distributions in the fluid domain. Most of the events concern thermally stratified flows, which result from differential heating (e.g. in heat exchangers) or from incomplete mixing of flows of different temperatures. For single-phase flows, stratified flow conditions develop in the case of PTS (see Section 6.2), hot-leg heterogeneities (Section 3.3), and thermal shock (Section 3.2). For two-phase flow problems, the reader is referred to the WG3 document [16]. Stratification may be one of the significant phenomena in the case of thermal shock under some SB-LOCA conditions and for condensation-induced waterhammer [79]. Stratification and buoyancy effects may lead to thermal fatigue, to modification of condensation rates, and to difficulties in predicting the associated mixing processes.

Stratified flows and buoyancy-induced effects take place in many parts of a NPP flow circuit: the RPV lower and upper plena, hot and cold legs, and secondary circuit pipework. In most instances, these phenomena are associated with unsteady 3-D flow situations. It is therefore necessary to derive a modelling strategy able to handle all the situations of relevance to nuclear reactor thermal hydraulics. In general, the phenomena are difficult to represent using a 1-D system-code approach, due to the geometric complexity. CFD is better suited in this regard, though it should be recognised that stable stratification may limit the action of turbulence mixing, while buoyancy may promote mixing, and the turbulent model in the CFD code must be able to reproduce these effects. For two-phase flows, the behaviour of the different phases of the flow, and the associated condensation rate, also need to be taken into account.

For the case of single-phase flows, there remain difficulties and uncertainties concerning the modelling of turbulence for such situations. The standard k-ε model [80] is known to poorly represent mixing in strongly buoyant situations, and more complex closures, for example, the Reynolds Stress Model [80], may need to be employed to more accurately reproduce the anisotropy of the turbulent fluctuations. Unfortunately, the RSM model is much less robust, and it may be difficult, or even impossible, to obtain converged solutions in complex geometries. Two further problems are that the transitional state of such flows is often difficult to predict, and the use of wall functions [80] may not be appropriate if they are not specifically designed for buoyant situations.

7.5 Fluid-Structure Interaction (CFD-FEM)

Flows in the primary circuit components of reactors are often strong enough to induce vibrations in, or damage to, confining or nearby structures, which may then have consequences regarding plant safety. In the case of thermal-hydraulics issues relating to the containment, there are instances of chugging and flow-induced condensation, producing jets in suppression pools in BWRs and associated mechanical loads on submerged structures. If the coupling is one-way (i.e. the structural motion does not have a feedback on the flow field), the computation is fairly straightforward, even under transient conditions. The velocity and temperature fields are first determined by the CFD module, and the thermal and mechanical loads are transferred via a data interface to the FEM (Finite Element Method) solver, from which the stresses in the solid structures are then evaluated.

However, in the case of two-way coupling, in which the structural deformation significantly alters the fluid parameters, such as in vibrational analysis, the CFD and FEM computations need to be performed simultaneously. This is expensive in terms of CPU time and often involves mesh reconstruction, which is also time consuming. There may also be problems in directly matching the CFD and FEM numerical algorithms. However, good progress is being made in this area, and in the commercial CFD world there are now strong corporate links between the CFD and FEM code vendors [37,44,45], so the technicalities of the coupling should soon become more automatic. Nonetheless, the assessment bases for fluid-structure interaction capability remain very problem-specific and need to be generalised in terms of generic examples, such as the oscillating cantilever, as described in the ERCOFTAC database [57].

7.6 Coupling of CFD Codes with Neutronics Codes

Precise predictions of the thermal loads to fuel rods and of core behaviour result from a balance between the thermal hydraulics and neutronics. The thermal hydraulics is coupled with the neutronics through the heat release due to neutronic activity (nuclear power generation), and the neutronics is coupled with the thermal hydraulics through the temperature (fuel and moderator), density (moderator), and the possible concentration of neutron absorber material (e.g. the boron concentration of the moderator). Only the nuclear community has an interest in these phenomena.

The current state-of-the-art is a coupling between a sub-channel description of the thermal hydraulics and neutron diffusion at the assembly level. However, some progress is being made in the direct coupling of CFD codes with existing neutronics packages: TRIO-U coupled to the Monte-Carlo neutronics code MCNP [81], FLICA4-3D and CAST3M coupled to CRONOS2 [82], and more recently a link between STAR-CD and CRONOS2 [83]. Possible improvements would be: (i) coupling of CFD codes with more advanced (i.e. deterministic or stochastic transport) neutronics models, (ii) development of a multi-scale approach to optimise the level of description with the conditions, since in many 3-D cases the power is very peaked (rod ejection, boron dilution, MSLB, etc.), and fine-scale models could be used only in a limited region, and (iii) development of time-step management procedures for complex transients in which the thermal hydraulics and neutronics time-scales are not the same.

Several benchmark exercises have been set up in the framework of OECD/NEA activities, including a PWR Main Steam Line Break, a BWR turbine trip, and for the VVER-1000 coolant transient (for which fine-mesh CFD models were used). However, a concerted effort is needed to bring together all appropriate data to place the assessment process on a sound basis.

7.7 Computing Power Limitations

The original version of *Parkinson's Law* [84]: "Work expands to fill the time available"; was first articulated by Prof. C. Northcote Parkinson in his book of the same name, and is based on an extensive study of the British Civil Service. The scientific observations, which contributed to the development of the law, included noting that as Britain's overseas Empire declined in importance, the number of employees at the Colonial Office correspondingly increased. The law was formulated in strict mathematical terms, but the work was so far ahead of its time that it was only 40 years later that the equations were finally vindicated. From this have arisen a number of variants. Two pertinent ones from the sphere of information technology are: *Parkinson's Law of Data*: "Data expands to fill the space available for storage", and *Parkinson's Law of Bandwidth Absorption:* "Network traffic expands to fill the available bandwidth". The application of CFD methodology also deserves a mention. Perhaps *Parkinson's Law of Computational Fluid Dynamics* could read: "The number of meshes expands to fill the available machine capacity". In reality, despite the overwhelming number of possibilities and advantages offered by present CFD codes, their role in nuclear reactor safety analyses remains limited. The development of codes able to compute LOCA phenomena with some realism began in the 1970s, which, by modern standards, was a period of very limited computing power. Typically, good turn-round could only be achieved using supercomputers. Today, a large part of the system calculations are carried out using workstations or PCs, and despite extended modelling capacity, the continuing upgrades in computer performance should ensure that system-code NRS analyses will never again require supercomputing power.

However, even with these advances in computer technology, it is difficult to see CFD codes being capable of simulating the entire primary or secondary loop of a nuclear plant for some time to come, so system and component codes will remain the main tools for computing system (and containment) behaviour in the near future. But, for those circumstances in which CFD is needed – and many examples of this have been alluded to in this paper – CFD computations will continue to stretch to the limit of available computing resources.

CFD simulations using 50 million nodes are now common in many industrial applications. However, in NRS applications, many of the situations requiring analysis are of a transient nature, they may be two-phase, and some of the transients are quite long. All CFD codes are, by nature, computationally demanding, both in terms of memory usage and in the number of operations. For a 3-D CFD simulation with N meshes in each coordinate direction, the total number of grid points is N^3. The time-step, though usually not CFL [85] restricted, remains for purely practical reasons roughly proportional to $1/N$, so the number of time steps is also proportional to N. Thus, the run-time for the CFD code should scale according to N^4, where the constant of proportionality, among other things, depends linearly on the total simulation time and, as remarked above, simulation times in NRS applications can be very long.

Speed-up can be achieved by partitioning the program to run on a number of processors in parallel. Since 1990, the use of parallel computation has shifted from being a marginal research activity to the mainstream of numerical computing. A recent study [86] has shown that the scaling up of performance with number of processors is strongly dependent on the size of the system arrays (for CFD, this translates directly into the number of meshes), as well as on the details of the particular computer architecture and memory hierarchy. The speed of a program also depends on the programming language (generally, Fortran is faster than C) and the compiler (levels of optimisation), but there are machine-dependent factors too. Generally speaking, modern workstations give good performance for small array sizes that fit into the processor's cache. If this is not possible, performance can drop dramatically.

Even for an ideal linear speedup, the N^4 dependence of runtime on number of meshes in one coordinate direction means that doubling the number of processors, and keeping total runtime the same, the number of meshes in each direction can only be increased by about 19%, say from 200 to 238. Conversely, doubling the mesh density, say from 200 to 400 in each coordinate direction, again keeping total runtime constant, means that the number of processors has to be increased by a factor of 16. Given these statistics, it is evident that the pursuit of quality and trust in the application of CFD to transient NRS problems, adhering strictly to the dictates of a Best Practice Guidelines philosophy of multi-mesh simulations [87], will stretch available computing power to the limit for some years to come. In the mid-term, compromises will have to be made: for example, examining mesh sensitivity for a restricted part of the computational domain or to a specific period in the full transient.

7.8 Scaling

A traditional scaling analysis for a validation test would consist of first normalising the conservation equations at the sub-system or component level for the test section, repeating this sub-system level scaling for all the components in the system, and then collecting the local scaling criteria into a set of overall system scaling criteria. The claim is then made that the dynamic component interaction and the global system response should be scaled successfully within the set of criteria for local component scaling, since the system is the sum of its components. This principle applies only if all the local criteria are met simultaneously. Except in the simplest (i.e. generic) cases, this ideal is physically impossible to achieve because the surface areas and volumes – and by inference, the area-dependent transfer rates and volume-dependent capacities – scale with different powers of the length parameter, thereby produce conflicting scaling requirements.

According to Wulff [88], two necessary conditions for successful scaling are: (i) the governing equations must be normalised such that the normalised variables (and their derivatives with respect to the normalised time and space coordinates) are of order unity, so that the magnitude of the normalised conservation equation can be measured by its normalising (constant) coefficient; and (ii) the governing equations must be scaled by dividing through by the coefficient of the driving term. This procedure brings the driving term to order unity and yields fewer non-dimensional scaling groups (which measure the magnitudes of their respective terms), since the importance of the associated transfer processes relative to the driving term, may be then judged in a hierarchical sense.

A categorisation of scaling approaches has been formulated by Yadigaroglu and Zeller [11]. The simplest scaling technique is linear scaling, in which all length ratios are preserved. According to this scaling strategy, the mass, momentum and energy equations of a system, along with the equation of state, are non-dimensionalised, and scaling criteria are then derived from the resulting parameters. The problem is then that linear length scaling inevitably leads to time distortion. As an alternative, volumetric or time-preserving scaling may be used instead. Such an approach is also based on scaling parameters derived from the non-dimensionalised conservation equations. Models scaled by these techniques preserve the flow lengths, while areas, volumes, flow rates, and power are reduced proportionally. Time-distorted scaling criteria, as described for example by Ishii and Kataoka [90], include both linear and volumetric scaling as special cases.

A “structured” scaling methodology, referred to as *hierarchical two-tiered scaling (H2TS)*, as proposed by Zuber [91], addresses the scaling issues from two fronts: a top-down (inductive) system approach, followed by a bottom-up (process-and-phenomena) approach. This strategy is proposed because the traditional local and component-level scaling strategies cannot reproduce the scaling criteria for component interaction. Altogether, the subject of scaling remains very unsure. For CFD applications to NRS, though the computational model can be performed at 1-1 scale, it is vital to ensure that the fluid-dynamic phenomena of relevance, validated against scaled experiments, have been preserved. If the fluid behaviour is categorised by flow-regime maps, it is essential that both the scaled validation test and the full-scale application lie in the same region of the map. This may be impossible to ensure for all phenomena simultaneously, whatever scaling strategy is followed. Thus, extrapolation to full size, whether it is a scale-up of a model facility or a CFD simulation, needs to be treated with great care to ensure the same physical phenomena are relevant at both scales.

8. NEW INITIATIVES

The WG2 Writing Group provided evidence to show that CFD is a tried-and-tested technology and that the main industrial-level CFD vendors were themselves taking active steps to quality-assure their software products by testing their codes against standard test data through active participation in international benchmark exercises. However, in a situation of low growth in the nuclear power industry, the primary driving forces for the development of CFD technology remain in non-nuclear areas, such as in the aerospace, automotive, marine, turbo-machinery, chemical, process industries, and to a lesser extent the environmental and biomedical industries. In the power-generation arena, the principal applications are again non-nuclear: combustion dynamics for fossil-fuel burning, gas turbines, vanes for wind turbines, etc.

Accepting the mandate to not only report on the existing assessment databases for the application of CFD to nuclear reactor safety issues but to also take steps to broaden and extend the databases, three new initiatives were instigated by the WG2 group:

- To organise a new series of international workshops to provide a forum for experimenters and numerical analysts to exchange information;
- To encourage nuclear departments at universities and research organisations to release test data by initiating international numerical benchmark exercises, and
- To establish a Wiki-type web portal that gives online access to the information collated by the group and documented in its final report; it also provides a means for updating and extending the information contained therein by inviting reader input.

The first of these activities was organised directly by the WG2 group, while the remaining two were accomplished by a smaller *Special CFD Group* that was formed later and consists of the chairmen of the three Writing Groups together with the NEA secretariat.

8.1 The CFD4NRS Workshop

The Workshop, entitled *CFD4NRS Benchmarking of CFD Codes for Application to Nuclear Reactor Safety*, was sponsored jointly by the OECD/NEA and IAEA and took place at Garching, Munich, Germany in September 2006. The Workshop provided a forum for both numerical analysts and experimenters to exchange information in the field of NRS-related activities relevant to CFD validation. Papers describing CFD simulations were accepted only if there was a strong validation component. Most related to NRS issues highlighted in this paper, such as pressurised thermal shock, boron dilution, hydrogen distribution, induced breaks, and thermal striping.

The use of Best Practice Guidelines [87] for the CFD simulations and the stipulation of error bounds and uncertainties on experimental measurements were both encouraged. Papers describing experiments that provided data suitable for CFD validation were strongly supported, a proviso being that *CFD-grade* data were available: for example, using LDA, hot-film/wire anemometry and Particle Image Velocimetry (PIV) for velocities and turbulence quantities, and Laser Induced Fluorescence (LIF) for species concentration. Papers describing experiments which only provided data in terms of integral measurements, i.e. area-averaged quantities, were not accepted. Though emphasis was placed on single-phase phenomena and separated flows, there was some scope for papers dealing with high-quality multi-phase flow experiments which featured local measurements of volume fractions and for multi-phase CFD validation exercises which followed BPGs. Figure 13 shows the frontispiece for the workshop flyer, showing the title and location of the workshop.

Fig. 13. Picture Taken from the Flyer Announcing the CFD4NRS Workshop

The case for future workshops in the series was discussed openly during the final panel session. It was noted that 2/3rds of the papers accepted for CFD4NRS were concerned with single-phase applications while 1/3rd were dedicated to multi-phase issues. The ratio, which probably reflects the degree of maturity of CFD in the respective areas, suggests a growing acknowledgement of the role of multi-phase CFD in nuclear safety issues. Selected papers from the workshop, including three from invited speakers, were subsequently included in a special issue of the journal Nuclear Engineering and Design [92].

Clear recommendations to emerge from the workshop for the continuing use of CFD methods in NRS issues are listed below.

- BPGs should be followed as far as practicable to ensure that CFD simulation results are free of numerical errors and that the physical models employed are well validated against data appropriate to the flow regimes and physical phenomena being investigated.
- Experimental data for CFD code validation should include estimates of measurement uncertainties and should include detailed information concerning initial and boundary conditions.
- Experimenters should collaborate actively with CFD practitioners in advance of setting up their instrumentation. Such communication is vital in ensuring that the information needed to set up the CFD simulation will actually be available, the selection of *target variables* (i.e. the most significant parameters against which to compare code predictions) is optimal, and the frequency of data acquisition is appropriate to the time-scale(s) of the most significant fluid-dynamic/heat-transfer/phase-exchange events.

The second workshop in the series, XCFD4NRS, took place in Grenoble, France in September 2008. Here, the emphasis was more on multi-phase aspects (see Table 1) and was centred around and organised by the WG3 Writing Group. Again, selected papers have been collected in a special issue of the journal Nuclear Engineering and Design, to appear shortly. The third workshop, CFD4NRS-3, will take place in Washington DC in September 2010, and plans are in place for a fourth workshop to take place in Daejeon, Korea in 2012.

8.2 Benchmark Exercise on Thermal Fatigue

During a meeting of the three Writing Group chairmen convened in Grenoble in September 2008 at the conclusion of the 2nd of the workshops, XCFD4NRS [93], discussions were held concerning candidate experiments around which to organise an international benchmark exercise; both single-phase and two-phase options were considered. It was generally acknowledged that it would be desirable to have the opportunity of performing a blind numerical simulation, and this would entail finding a completed experiment for which the data had not yet been released or encouraging a new experiment (most likely in an existing facility) to be undertaken especially for this exercise. The group took on the responsibility of finding a suitable experiment, for providing the organisational basis for launching the benchmark exercise (though not on the scale of an International Standard Problem, ISP), and for the synthesis of the results.

An early opportunity came in the area of thermal fatigue near a T-junction. As discussed in Sections 3.2 and 6.3 of this paper, failures of structures due to high-cycle thermal fatigue have occurred in several nuclear plants around the world in different reactor types, usually associated with mixing zones where hot and cold streams meet, particularly downstream of T-junctions. In addition, in a recent PIRT-type study [17], the issue of thermal fatigue came moderately high on the list of priority single-phase NRS issues from Table 1, and there appeared to be a good degree of conformity of interest internationally. Figure 14 is a schematic of the geometric situation in a typical T-junction configuration, showing the turbulent mixing zone downstream of the junction and typical pipe locations where cracks may be expected to appear due to thermal fatigue.

Tests on thermal mixing in a T-junction were being performed in 2007 at the Älvkarleby Laboratory of Vattenfall Research and Development, with the primary aim of providing high-quality validation data for CFD calculations [94]. The test section (Fig. 11) is constructed from Plexiglas and the junction itself from one solid block into which the main and branch pipes fit. The temperatures of the water in the main and branch pipes are maintained at about 15ºC and 30ºC, respectively, with minimal heat loss from exposed surfaces. Special care was taken to provide simple and well-defined inlet boundary conditions to remove ambiguities in defining the accompanying CFD input data.

In these experiments, temperature fluctuations near pipe walls were measured using thermocouples. These were placed around the inner wall perimeter of the main pipe, at seven stations downstream of the junction, and at one station upstream. Velocity profiles upstream and downstream of the junction were measured using a two-component LDA system. Data are available in the form of mean and root-mean-square values. Following negotiations with representatives from Vattenfall, it was agreed to perform a special mixing test in the series and keep the data secret to provide a basis for a blind benchmark exercise.

The benchmark was launched at a kick-off meeting in May 2009; at which time the official, detailed benchmark specifications were released, and groups were invited to submit CFD simulation results about one year later. The leader of the Vattenfall experimental team was invited to join the benchmark organising committee. Interest in the activity was expressed by 65 groups around the world, and of these 29 submitted blind simulation data for synthesis. Results are to be presented at the CFD4NRS-3 Workshop in Washington DC in September 2010.

8.3 Construction of the CFD for NRS Wiki Page

The activities of the three OECD/NEA Writing Groups on CFD were concluded at the end of 2007 with the completion of their respective CSNI reports. Like any state-of-the-art report, these documents are only up-to-date at the time of writing, and, given the rapidly expanding use of CFD as a refined analysis tool in nuclear technology, the information they contain will soon become outdated. To preserve their topicality, improvements and extensions to the documents are foreseen. It was decided that the most efficient vehicle for regular updating would be to create a Wiki-type web portal. Consequently, in a pilot study, a dedicated webpage has been created on the NEA website using Wikimedia software [95]. In a first step,

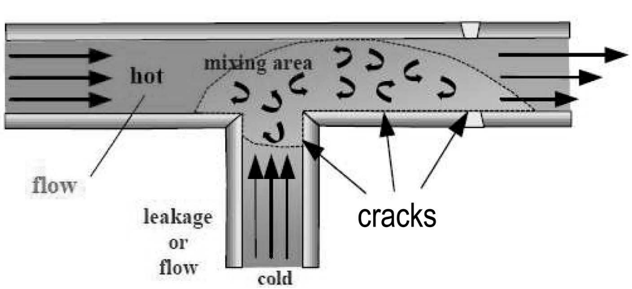


Fig. 14. Typical T-junction Configuration Showing Zone of Turbulent Mixing and Possible Fatigue Crack Locations

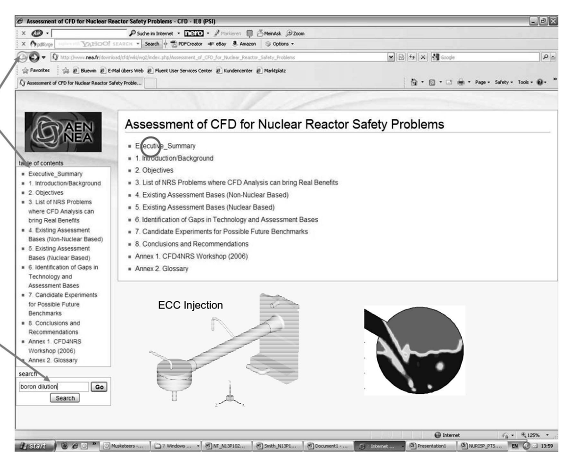


Fig. 15. Current Layout of the WG2 Main Wiki Page

the WG2 document in the form it appears in the archival document [15] has been uploaded, to provide on-line access. (The WG1 document [87] has since also been uploaded, and the webpages for the WG3 document [16] are under construction.)

Figure 15 shows the current version of the main page. Listed are the main chapter headings of the WG2 document, each being an active internal link to the actual detailed information. For example, clicking on the item *Executive Summary* (circled) opens up the pages containing the Executive Summary in its entirety, just as it exists in the original archival document. There is also an active scroll bar, and a multi-level search facility. Navigation can be via the Navigation Bar or by use of the Internet Browser function, as indicated.

The larger chapters are subdivided; clicking on the chapter heading leads to a page containing the sub-division headings. These are themselves active links, and clicking here leads directly to the documented material. Active links are also being installed at this level, to enable the user to navigate quickly to other parts of the document. The web-page addresses, for example to the commercial CFD sites, are also active, and it is planned to install a similar facility for the journal references, which will be useful for registered subscribers with electronic access to the material.

However, the most useful feature of the web portal will be the opportunity to modify, correct, update and extend the information contained therein. The Wiki concept is the vehicle for this. The aim is to have a static site, with unrestricted access. Readers will not be able to directly edit or change the information there but can communicate their suggestions to the website editors. In parallel, a beta version of the webpage will be maintained for installing updates prior to transfer to the static site. At present, access to the beta version is restricted to the three former chairmen of the OECD/NEA Writing Groups, who have editing responsibility for the website versions of their respective documents, together with the NEA webmaster. It will be the editor's responsibility to review all new submissions and implement them into the open-access version of the site, following approval from the CSNI. This responsibility is now being extended to a full OECD/NEA scientific committee, since the burden of work is expected to become excessive.

9. CONCLUSIONS

The use of computational methods for performing safety analyses of reactor systems has been established for more than 30 years. Very reliable codes have been developed for analysing the pipework and components of the primary system, and results from these analyses are often used in the safety assessment of nuclear power systems undertaken by the regulatory authorities. Such codes are based on networks of 1-D or even 0-D cells. However, the flow in many reactor primary components is essentially 3-D in nature, as is natural circulation, mixing and stratification in containments. CFD has the potential to numerically simulate flows of this type and to handle geometries of almost arbitrary complexity. Already, CFD is being applied to these and similar flow situations to better quantify safety margins, and it is expected to feature more prominently in reactor safety analyses in the future.

The traditional approaches to nuclear reactor safety (NRS) analysis, using system codes for example, take advantage of the very large database of mass, momentum, and energy exchange correlations that have been built into them. The correlations have been formulated from essentially 1-D special-effects experiments, and their ranges of validity are well known and controlled internally within the numerical models. Notwithstanding the scaling issues which still need to be resolved, herein lies the trustworthiness of the numerical predictions of the system codes. Analogous databases for 3-D flows are very sparse by comparison, and the issue of the trust and reliability of CFD codes for use in nuclear reactor safety applications has to be addressed before the use of CFD can be considered at a similar level. This issue represented the primary focus of the work carried out by the WGAMA CFD Writing Group WG2. A summary of its findings has been embodied in this article.

A list of NRS problems for which CFD analysis is considered to result in positive benefits has been compiled. The list contains safety issues of relevance to fluid flows in the core, the primary circuit, and containment, under normal or abnormal operating conditions and during accident sequences. The list contains single-phase and two-phase flow examples, though in the latter case reference is made to the document dealing with the *Extension of CFD Codes to Two-Phase Flow Nuclear Reactor Safety Problems*, which accompanies this paper.

Recognising that CFD is already an established technology outside the nuclear domain, a list of the existing assessment bases from other application areas has been drawn up by the WG2 group; some highlights are given here. It is shown that these databases are principally of two types: those concerned with general aspects of CFD validation, such as ERCOFTAC, and those focussed on specialised topics; for example, NPARC and AIAA. In addition, most CFD codes currently being used for NRS analysis have their own, custom-built assessment bases; the data is provided from both within and externally to the nuclear community. It was concluded that application of CFD to NRS problems can benefit indirectly from these databases, since many of the thermal hydraulic situations are of a similar character.

Certainly more focussed on NRS issues are the validation experiments carried out specifically to address safety issues within the nuclear technology field; these have also been listed with evaluations of their usefulness. Typical examples are experiments devoted to the boron dilution and in-vessel mixing issues, pressurised thermal shock, and thermal fatigue in pipes, all of which have already been the subject of previous benchmarking activities.

The technology gaps which need to be closed to make CFD a more qualified numerical tool have also been identified. These include lack of appropriate validation data (aerosol deposition in containments), limitations in the range of application of turbulence models (for example in stratified and buoyant flows), coupling of CFD with neutronics, system and structural mechanics codes, and the need to keep simulations at a manageable size due to computer power limitations in simulating long transients. It was noted that good progress is being made in closing most of these gaps.

Important new information has been provided by the material presented at the CFD4NRS Workshop, during which CFD-grade data from experiments and numerical simulations with a strong emphasis on validation were presented. The workshop forum also engendered ideas on new benchmarking activities, and an international benchmarking activity in the area of high-cycle thermal fatigue was subsequently launched. In total, 29 groups submitted "blind" CFD simulations of a mixing tee experiment performed by Vattenfall, Sweden. A synthesis will be presented at the 3rd workshop in the series (Sept. 2010).

CFD is a very dynamic technology, and with its increasing use within the nuclear domain there will be ever greater demands to document current capabilities and demonstrate quality and trust by means of validation exercises. It is expected therefore that any catalogue of the assessment databases relevant to NRS will expand to keep pace with the software development. To prevent the important information assembled by the Writing Groups from becoming obsolete, a web-based centre to consolidate, update, and extend the information, based on Wiki software, has been set up on the NEA website. This portal will ensure that existing and future NRS benchmarking activities will be as up to date as possible and readily accessible to those who need them.

ACKNOWLEDGMENTS

The author is grateful to the members of the WG2 group for the concerted effort they made in compiling the CFD assessment base referred to in this paper. In alphabetical order, thanks are due to: D. Bestion, U. Bieder, E. Graffard, M. Heitsch, M. Henriksson, T. Höhne, E. Komen, J. Mahaffy, J. Macek, F. Moretti, T. Morii, P. Mühlbauer, U. Rohde, M. Scheuerer, C.-H. Song, G. Zigh.

REFERENCES

[1] Harlow, F.H., Amsden, A.A., "Flow of Interpenetrating Material Phases." *J. Comp. Phys.*, **18**, 440-464 (1975).

[2] RELAP5/Mod3 code manual – code structure, system models and solution methods, Vol. I., The Thermal Hydraulics Group, SCIENTECH Inc., Idaho, June 1999.

[3] TRACE V5.0 theory manual – field equations, solution methods and physical models, Office of Nuclear Regulatory Research. US NRC, 2007.

[4] Bestion, D., Barre, F., Faydide, B., "Methodology, status and plans for development and assessment of the CATHARE code", Proc. OECD/CSNI Int. Conf., Annapolis, USA, 5-8 Nov. 1999.

[5] Graf U., "Implicit Coupling of Fluid-Dynamic Systems: Application to Multidimensional Countercurrent Two-Phase Flow of Water and Steam", *Nucl. Sci. Eng.*, **129**, 305-310 (1998).

[6] GOTHIC Containment Analysis Program, Version 7.2a(QA), EPRI, Palo Alto, CA, Jan. 2006.

[7] J. R. Travis, et al., "GASFLOW: A Computational Fluid Dynamics Code for Gases, Aerosols, and Combustion, Volume 2, User's Manual", LA-13357-MS, FZKA-5994, FZK Karlesruhe, 1998.

[8] Gauntt, R.O. et al., MELCOR Computer Code Manuals, Version 1.8.5, May 2000, NUREG/CR-6119, SAND-2417/2, May 2000.

[9] Siefken, L.J., Coryell, E.W., Harvego, E.A., Hohorst, J.K., SCDAP/RELAP5/MOD3.3 Code Manual, NUREG/CR-6150, INEL-96/0422, January 2000.

[10] http://www.fauske.com/maap.html.

[11] Yadigaroglu, G., Zeller, G.M., "Fluid-to-fluid scaling for a gravity- and flashing-driven natural circulation loop", *Nucl. Eng. Des*., **151**, 49–64 (1994).

[12] D'Auria, F., "Overview of Uncertainty Methods in Nuclear Reactor Thermal-Hydraulics", Invited Paper N13P1446, *Proc. 13th Int. Top. Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-13)*, Kanazawa City, Japan, Sept. 27 – Oct. 2, 2009.

[13] http://www.opencfd.co.uk/openfoam/

[14] CSNI Report NEA/CSNI/R(2002)16, OECD Nuclear Energy Agency, Paris, France, 2002.

[15] Smith, B.L. (ed.), "Assessment of Computational Fluid Dynamics (CFD) for Nuclear Reactor Safety Problems", OECD Nuclear Energy Agency, Technical Report, NEA/CSNI/R(2007)13, Jan. 2008.

[16] Bestion, D. (ed.) "Extension of CFD Codes to Two-Phase Flow Safety Problems", OECD, Nuclear Energy Agency, Technical Report, NEA/CSNI/R(2010)2, March 2010.

[17] Smith, B.L. "Identification and Prioritization of Generic Nuclear Safety Problems Requiring CFD Analysis", Paper 75482, *Proc. 17th Int. Conf. on Nuclear Engineering (ICONE-17)*, Brussels, Belgium, July 12-16, 2009.

[18] Queral, C., Gonzalez, I., "Analysis of Heterogeneous Boron Dilution Sequences", *Proc. Int. Conf: Nuclear Energy for New Europe 2004*, Portorož, Slovenia, Sept. 6-9, 2004.

[19] Gavrilas, M., Kiger, K., "ISP-43: Rapid Boron Dilution Transient Experiment. Comparison Report", OECD Nuclear Energy Agency, Technical Report, NEA/CSNI/R(2000)22, Feb. 2001.

[20] Hoehne, T., Prasser, H.-M., Rohde, U., "Numerical coolant mixing in comparison with experi-ments at the ROCOM test facility", *Proc. ANS Winter Meeting*, Reno, USA, Nov. 11-15, 2001

[21] Hemström, B., Karlsson, R., Henriksson, M., "Experiments and Numerical Modelling of Rapid Boron Dilution Transients in a Westinghouse PWR", *Annual Meeting on Nuclear Technology*, Berlin, Germany, May 20-22, 2003.
[22] Tuomisto, H. *et al*., "EUBORA - Concerted Action on Boron Dilution Experiments", *Proc. FISA-99 Symp*., Luxembourg, 29 Nov. - 1 Dec., 1999.
[23] Weiss, F.-P. *et al*., "Fluid Mixing and Flow Distribution in the Reactor Circuit (FLOWMIX-R)", *Proc. FISA-2003, Symp*., Luxembourg, 10-13 Nov., 2003.
[24] Wakamatsu, M., Nei, H., Hashiguchi, K., "Attenuation of Temperature Fluctuations in Thermal Striping", *J. Nucl. Sci. Tech*., **32**(8), 752-762 (1995).
[25] Jungclaus, D., *et al*., "Common IPSN/GRS Safety Assessment of Primary Coolant Unisolable Leak Incidents caused by Stress Cycling", NEA/CSNI Specialist Meeting on Experience with Thermal Fatigue in LWR Piping caused by Mixing and Stratification, OECD/NEA, Paris France, 7-12 June, 1998.
[26] Shah V.N., *et al*., "Assessment of Field Experience Related to Pressurized Water Reactor Primary System Leaks", *Proc. ASME PVP Conf*., Boston, USA, Aug. 1-5, 1999.
[27] Chapuliot, S. *et al*., "Hydro-thermal-mechanical analysis of thermal fatigue in a mixing tee", *Nucl. Eng. Des*., **235**, 575-596 (2005).
[28] Hoehne, T., Kliem, S., Bieder, U., "Modeling of a buoyancy-driven flow experiment at the ROCOM test facility using the CFD-codes CFX-5 and TRIO_U", *Nucl. Eng. Des*., **236**(12), 1309-1325 (2006).
[29] Shiina, K. et al., "Evaluation of Flow Characteristics in the Lowere Plenum of the ABWR by using CFD Analysis", Paper 36393, *Proc. 11th Int. Conf. on Nuclear Engineering (ICONE-11)*, Tokyo, Japan, April 20-23, 2003.
[30] Tinoco, H., "Three-Dimensional Modelling of a Steam-Line Break in a Boiling Water Reactor", *Nucl. Safety and Eng*., **140**, 152-164 (2002).
[31] Wolf, L., "Experimental results of coupled fluid-structure interaction during blow down of the HDR-vessel and comparison with pre- and post-test prediction", *Nucl. Eng. Des*., **70**, 269-308 (1982).
[32] Karwat, H. (ed.), "SOAR on Containment Thermal Hydraulics and Hydrogen Distribution", OECD Nuclear Energy Agency, Technical Report, NEA/CSNI/R(1999)16,
[33] Brun, P. *et al*., "Specification of International Standard Problem on Containment Thermal-Hydraulics ISP-47, Step 1: TOSQAN–MISTRA," Institut de Radioprotection et de Sureté Nucléaire (IRSN) and Comissariat à l'Energie Atomique (CEA), Saclay, France, 2002.
[34] Caron-Charles, M., Quillico, J.J.. Brinster, J., "Steam condensation experiments by the MISTRA facility for field containment code validation", Paper 22661, *Proc. 10th Int. Conf. on Nucl. Eng. (ICONE-10)*, Arlington, Virginia, USA, April 14-18, 2002
[35] Sonnenkalb, M., Poss, G., "The International Test Programme in the THAI Facility and its Use for Code Validation", *Eurosafe 2009*, Brussels, Belgium, Nov. 2-3, 2009.
[36] Paladino, D., Andreani, M., Zboray, R., Dreier, J., "Toward a CFD-Grade Database Addressing LWR Containment Phenomena", *Proc. OECD/NEA-IAEA Joint Conf. on CFD for Nuclear Reactor Safety Applications (CFD4NRS-3)*, Bethesda, Maryland, USA, Sept. 14-16, 2010.
[37] http://www.fluent.com/
[38] Redlinger, R., "DET3D - A CFD Tool for Simulating Hydrogen Combustion in Nuclear Reactor Safety, *Proc. CFD4NRS: Conf. on Benchmarking of CFD Codes for Applications to Nuclear Reactor Safety*,", Garching, Munich, Germany, 5-7 Sept. 2006.
[39] http://www.abaqus.com
[40] Dorofeev, S.B., et al., "Large scale hydrogen/air detonation experiments: the effect of ignition location and hydrogen concentration on load", Report number RRCKI- 80e05/59, Russian Research Center, Kurchatov Institute, 1993.
[41] Breitung W, et al., "Integral large-scale experiments on hydrogen combustion for severe accident code validation-HYCOM", *Nucl. Eng. Des*., **70**, 235-253 (2005).
[42] Bielert, U., et al., "Multi-dimensional simulation of hydrogen distribution and turbulent combustion in severe accidents", *Nucl. Eng. Des*., **209**, 165-172 (2001).
[43] Hardt, P. von der, Jones, A.V., Lecomte., C., Tattegrain, A., "The PHEBUS FP Severe Accident Experimental Programme", *Nuclear Safety*, **35**(2), 187-205 (1994).
[44] http://www.ansys.com/
[45] http://www.ansys.com/Products/cfx.asp
[46] http://www.simulia.com/products/abaqus_cfd.html
[47] Oberkampf, W., Trucano, T.G., "Verification and Validation Benchmarks", *Nucl. Eng. Des*., **238**(3), 716-743 (2008).
[48] Roache, P.J., "Code Verification by the Method of Manufactured Solutions", *ASME J. Fluids Eng*., **124**(1), 4-10 (2002)
[49] Roache, P.J., *Verification and Validation in Computational Science and Engineering*, Hermosa Publishers, Albuquerque, New Mexico, 1998.
[50] Wilson, G.E., Boyack, B.E., "The role of the PIRT process in experiments, code development and code applications associated with reactor safety analysis", *Nucl. Eng. Des*., **186**, 23-37 (1998)
[51] http://www.cd-adapco.com/
[52] http://www.cham.co.uk/
[53] http://www-trio-u.cea.fr/
[54] Méchitoua, M., *et al*., "An unstructured finite volume solver for two-phase water-vapour flows based on an elliptic oriented fractional step method," *Proc. 11th Int. Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH 11)*, Avignon, France, Oct. 2-6, 2005.
[55] http://www.opencfd.co.uk/openfoam/
[56] http://www.esi-group.com/products/Fluid-Dynamics/cfd-ace
[57] http://www.ercoftac.org/
[58] Vos, J.B., *et al*., "Overview of Application Challenges in the Aeronautical Industry", QNET-CFD Network Bulletin, No. 1, pp. 1-7, July 2001.
[59] http://www.grc.nasa.gov/WWW/wind/valid/validation.html/
[60] Hoehne, T., Prasser, H.-M., Rohde, M., "Numerical coolant mixing in comparison with experiments at the ROCOM test facility", *Proc. ANS Winter Meeting*, Reno, USA, Nov. 11-15, 2001
[61] Prasser, H.-M., *et al*., "Coolant mixing in a PWR – deboration transients, steam line breaks and emergency core cooling injection – experiments and analyses", *Nucl. Technol*., **143** (1), 37–56 (2003)
[62] Theofanous, T.G., Yan, H., "A unified interpretation of

one-fifth to full-scale thermal mixing experiments related to pressurized thermal shock", NUREG/CR-5677, April 1991.

[63] Wolf, L., "Experimental results of coupled fluid-structure interaction during blowdown of the HDR-vessel and comparison with pre- and post-test prediction", *Nucl. Eng. Des*., **70**, 269-308 (1982).

[64] Scheuerer, M., *et al*., "Evaluation of computational fluid dynamic methods for reactor safety analysis (ECORA)", *Nucl. Eng. Des*., **235**, 359-368 (2005).

[65] Prasser, H.-M., *et al*., "The multipurpose thermal hydraulic test facility TOPFLOW: an overview on experimental capabilities, instrumentation and results", *Nucl. Eng. Des*., **237**, 1848-1861 (2007).

[66] http://www.nuresim.com/

[67] Metzner, K.-J., Wilke, U., "European THERFAT project – thermal fatigue evaluation of piping system "Tee"-connections", *Nucl. Eng. Des*., **235**, 473-484 (2005).

[68] Zboray, R., Manera, A., Niceno, B., Prasser H.-M., "Investigations on Mixing Phenomena in Single-Phase Flows in a T-Junction Geometry", *Proc. 12th Int. Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-12)*, Paper No. 71, Pittsburgh, USA, Sept. 30 - Oct. 4, 2007

[69] http://www.nea.fr/nsd/csni/cfd/

[70] Jeong, J.J., et al., "Assessment of the COBRA/RELAP5 Code Using the LOFT l2-3 Large-Break Loss-of-Coolant Experiment", *Ann. Nucl. Energy*, **24**, 1171-1182 (1997).

[71] Graf, U., "Implicit Coupling of Fluid-Dynamic Systems: Application to Multidimensional Countercurrent Two-Phase Flow of Water and Steam", *Nucl. Sci. Eng*., **129**, 305-310 (1998).

[72] Kliem, S., Hoehne, T., Rohde, U., Weiss, F.-P., "Main Steam Line Break Analysis of a VVER-440 Reactor Using the Coupled Thermohydraulics System/3D-Neutron Kinetics Code DYN3D/ATHLET in Combination with the CFD Code CFX-4", *Proc. 9th Int. Top. Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-9)*, San Francisco, USA, Oct. 3-8, 1999.

[73] Aumiller, D.L., Tomlinson, E.T., Weaver, W.L., "An Integrated RELAP5-3D and Multiphase CFD Code System Utilizing a Semi-Implicit Coupling Technique", *Nucl. Eng. Des*., **216**, 77-87 (2002).

[74] Schultz, R.R., Weaver, W.L., Ougouag, A.M., "Validating & verifying a new thermal-hydraulic analysis tool", *Proc. 10th Int. Conf. on Nuclear Engineering (ICONE10)*, Arlington, USA, April 14-18, 2002.

[75] Bertolotto, D., *et al*., "Single-phase mixing studies by means of a directly coupled CFD/ system-code tool", *Ann. Nucl. Energy*, **36**(3), 310-316 (2009).

[76] Firnhaber, M., Kanzleiter, T.F., Schwarz, S., Weber, G., "International Standard Problem ISP-37. VANAM M3 – A multi compartment aerosol depletion test with hygroscopic aerosol material", Comparison Report, OCDE/GD(97)16, December 1996.

[77] Makynen, J., Jokiniemi, J., (eds.), "CSNI/PWG4/FPC AHMED Code Comparison Exercise", OECD Nuclear Energy Agency, Technical Report, NEA/CSNI/R(95)23, October 1995.

[78] Firnhaber, M., Fischer, K., Schwarz, S., Weber, G., "International Standard Problem ISP-44 KAEVER. Experiments on the behaviour of core-melt aerosols in a LWR containment". OECD Nuclear Energy Agency, Technical Report, NEA/CSNI/R(2003)5.

[79] Prasser, H.-M., Boettger, A., Zschau, J., Baranyai, G., Ezsoel, Gy., "Thermal Effects During Condensation Induced Water Hammer Behind Fast Acting Valves In Pipelines", Paper 36310, *Proc. 11th Int, Conf. on Nucl. Eng. (ICONE-11)*, Shinjuku, Tokyo, Japan, 20-23 April, 2003.

[80] Rodi, W., *Turbulence Models and their Application in Hydraulics: A State of the Art Review*, IAHR Monograph, 3rd edition, A. A. Balkema, Rotterdam, 2000.

[81] Perdu, F., "Contributions aux études de sûreté pour des filières innovantes de réacteurs nucléaires", PhD Thesis, Université Joseph Fourier, Grenoble, 2003.

[82] Studer, E., *et al*., "Gas Cooled Reactor Thermal-Hydraulics using the CAST3M and CRONOS2 codes", *Proc. 9th Int. Top. Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-10)*, Seoul, Korea, Oct. 5-9, 2003.

[83] Billot,P., Hittner, D., Vasseur, P., "Outlines of the French R&D Programme for the Development of High and Very High Temperature Reactors", *3rd Int. Topical Meeting on High Temperature Reactor Technology*, Johannesburg, S.A., Oct. 1-4, 2006.

[84] Parkinson, C.N., *Parkinson's Law: The Pursuit of Progress*, London, John Murray, 1958.

[85] Peyret, R., Taylor, Th., *Computational Methods for Fluid Flow*, p42, Springer-Verlag, New York, 1983.

[86] Norris, S.E., "A Parallel Navier-Stokes Solver for Natural Convection and Free-Surface Flow", Ch. 6, PhD Thesis, Dept. Mech. Eng., University of Sydney, Sept. 2000.

[87] Mahaffy, J. (ed.) "Best Practice Guidelines for the Use of CFD in Nuclear Reactor Safety Applications", OECD, Nuclear Energy Agency, Technical Report, CSNI/R(2007)5, April 2007

[88] Wulff, W., "Scaling of thermohydraulic systems", *Nucl. Eng. Des*., **163**, 359–395 (1996).

[89] Yadigaroglu, G., Zeller, M., "Fluid-to-fluid scaling for a gravity - and flashing-driven natural circulation loop", *Nucl. Eng. Des*., **151**, 49–64 (1994).

[90] Ishii, M., Kataoka, I., "Scaling laws for thermal-hydraulic system under single phase and two-phase natural circulation", *Nucl. Eng. Des*., **81**, 411–425 (1984).

[91] Zuber, N., *et al*., "An integrated structure and scaling methodology for severe accident technical issue resolution: development of methodology", *Nucl. Eng. Des*., **186**, 1–21 (1998).

[92] Smith, B. L., Hassan, Y. (eds.) "CFD4NRS: Benchmarking of CFD Codes for Application to Nuclear Reactor Safety", *Special Issue, Nucl. Eng. Des*., **238**(3), 443-785 (2008).

[93] XCFD4NRS: Experiments and CFD Code Applications to Nuclear Reactor Safety, Grenoble, France, 10-12 Sept. 2010 (CD-ROM).

[94] Westin, J. et al., "High-cycle thermal fatigue in mixing tees. Large eddy simulations compared to a new validation experiment", Paper 48731, *Proc. 16th Int. Conf. on Nucl. Engng (ICONE16)*, Orlando, Florida, USA, May 11-15, 2008.

[95] http://meta.wikimedia.org/wiki/Help:Introduction/