Nuclear Safety
NEA/CSNI/R(2014)12
January 2015
www.oecd-nea.org

Assessment of CFD Codes for Nuclear Reactor Safety Problems – Revision 2

BETTER POLICIES FOR BETTER LIVES
NUCLEAR ENERGY AGENCY

Unclassified **NEA/CSNI/R(2014)12**

Organisation de Coopération et de Développement Économiques
Organisation for Economic Co-operation and Development

16-Jan-2015

English text only

NUCLEAR ENERGY AGENCY
COMMITTEE ON THE SAFETY OF NUCLEAR INSTALLATIONS

NEA/CSNI/R(2014)12
Unclassified

Assessment of CFD Codes for Nuclear Reactor Safety Problems - Revision 2

JT03369378

English text only

ORGANISATION FOR ECONOMIC CO-OPERATION AND DEVELOPMENT

The OECD is a unique forum where the governments of 34 democracies work together to address the economic, social and environmental challenges of globalisation. The OECD is also at the forefront of efforts to understand and to help governments respond to new developments and concerns, such as corporate governance, the information economy and the challenges of an ageing population. The Organisation provides a setting where governments can compare policy experiences, seek answers to common problems, identify good practice and work to co-ordinate domestic and international policies.

The OECD member countries are: Australia, Austria, Belgium, Canada, Chile, the Czech Republic, Denmark, Estonia, Finland, France, Germany, Greece, Hungary, Iceland, Ireland, Israel, Italy, Japan, Luxembourg, Mexico, the Netherlands, New Zealand, Norway, Poland, Portugal, the Republic of Korea, the Slovak Republic, Slovenia, Spain, Sweden, Switzerland, Turkey, the United Kingdom and the United States. The European Commission takes part in the work of the OECD.

OECD Publishing disseminates widely the results of the Organisation’s statistics gathering and research on economic, social and environmental issues, as well as the conventions, guidelines and standards agreed by its members.

NUCLEAR ENERGY AGENCY

The OECD Nuclear Energy Agency (NEA) was established on 1 February 1958. Current NEA membership consists of 31 countries: Australia, Austria, Belgium, Canada, the Czech Republic, Denmark, Finland, France, Germany, Greece, Hungary, Iceland, Ireland, Italy, Japan, Luxembourg, Mexico, the Netherlands, Norway, Poland, Portugal, the Republic of Korea, the Russian Federation, the Slovak Republic, Slovenia, Spain, Sweden, Switzerland, Turkey, the United Kingdom and the United States. The European Commission also takes part in the work of the Agency.

The mission of the NEA is:

– to assist its member countries in maintaining and further developing, through international co-operation, the scientific, technological and legal bases required for a safe, environmentally friendly and economical use of nuclear energy for peaceful purposes, as well as
– to provide authoritative assessments and to forge common understandings on key issues, as input to government decisions on nuclear energy policy and to broader OECD policy analyses in areas such as energy and sustainable development.

Specific areas of competence of the NEA include the safety and regulation of nuclear activities, radioactive waste management, radiological protection, nuclear science, economic and technical analyses of the nuclear fuel cycle, nuclear law and liability, and public information.

The NEA Data Bank provides nuclear data and computer program services for participating countries. In these and related tasks, the NEA works in close collaboration with the International Atomic Energy Agency in Vienna, with which it has a Co-operation Agreement, as well as with other international organisations in the nuclear field.

COMMITTEE ON THE SAFETY OF NUCLEAR INSTALLATIONS

Within the OECD framework, the NEA Committee on the Safety of Nuclear Installations (CSNI) is an international committee made of senior scientists and engineers, with broad responsibilities for safety technology and research programmes, as well as representatives from regulatory authorities. It was set up in 1973 to develop and co-ordinate the activities of the NEA concerning the technical aspects of the design, construction and operation of nuclear installations insofar as they affect the safety of such installations.

The committee's purpose is to foster international co-operation in nuclear safety amongst the NEA member countries. The CSNI's main tasks are to exchange technical information and to promote collaboration between research, development, engineering and regulatory organisations; to review operating experience and the state of knowledge on selected topics of nuclear safety technology and safety assessment; to initiate and conduct programmes to overcome discrepancies, develop improvements and research consensus on technical issues; and to promote the co-ordination of work that serves to maintain competence in nuclear safety matters, including the establishment of joint undertakings.

The clear priority of the committee is on the safety of nuclear installations and the design and construction of new reactors and installations. For advanced reactor designs the committee provides a forum for improving safety related knowledge and a vehicle for joint research.

In implementing its programme, the CSNI establishes co-operative mechanisms with the NEA's Committee on Nuclear Regulatory Activities (CNRA) which is responsible for the programme of the Agency concerning the regulation, licensing and inspection of nuclear installations with regard to safety. It also co-operates with the other NEA's Standing Committees as well as with key international organizations (e.g., the IAEA) on matters of common interest.

ASSESSMENT OF CFD FOR NUCLEAR REACTOR SAFETY PROBLEMS

B. L. Smith (PSI), M. Andreani (PSI), U. Bieder (CEA), F. Ducros (CEA), E. Graffard (IRSN),
M. Heitsch (GRS), M. Henriksson (Vattenfall), T. Höhne (FZD), M. Houkema (NRG),
E. Komen (NRG), J. Mahaffy (PSU), F. Menter (ANSYS), F. Moretti (UPisa),
T. Morii (JNES), P. Mühlbauer (NRI), U. Rohde (HZDR), M. Scheuerer (GRS),
C.-H. Song (KAERI), T. Watanabe (JAEA), G. Zigh (US NRC)

With additional input from

F. Archambeau (EDF), S. Bellet (EDF), D. Bestion (CEA), C. F. Boyd (US NRC), E. Krepper (HZDR),
J.M. Muñoz-Cobo (UPV), J.-P. Simoneau (AREVA)

EXECUTIVE SUMMARY

Original Initiative

Following recommendations made at 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", held in Aix-en-Provence, France, 15-16 May, 2002, and a follow-up meeting "Use of Computational Fluid Dynamics (CFD) Codes for Safety Analysis of Reactor Systems including Containment", which took place in Pisa on 11-14 Nov., 2002, a CSNI action plan was drawn up which resulted in the creation of three Writing Groups, with mandates to perform the following tasks:

(1) Provide a set of guidelines for the application of CFD to NRS problems;

(2) Evaluate the existing CFD assessment bases, and identify gaps that need to be filled;

(3) Summarise the extensions needed to CFD codes for application to two-phase NRS problems.

Work began early in 2003. In the case of Writing Group 2 (WG2), a preliminary report was submitted to WGAMA[#] in September 2004 that scoped the work needed to be carried out to fulfil its mandate, and made recommendations on how to achieve the objective. A similar procedure was followed by the other two groups, and in January 2005 all three groups were reformed to carry out their respective tasks. In the case of WG2, this resulted in the issue of a CSNI report (NEA/CSNI/R(2007)13), issued in January 2008, describing the work undertaken.

Background

Computational methods have been used in the safety analysis of reactor systems for nearly 40 years. During this time, very reliable numerical programs have been developed for analysing the primary system, and similar programs have also been written for modelling containments and severe accident scenarios. Such codes model the reactor components as networks of 1-D or even 0-D cells. It is evident, however, that the flows in many reactor primary components are essentially 3-D in character, 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. Consequently, CFD is expected to feature more prominently in reactor thermal-hydraulics analyses in the future.

Traditional approaches to NRS analysis, using system codes for example, have been successful because 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 tests, and their specific ranges of validity have been very well scrutinised. Analogous data relating to 3-D flow situations is very sparse by comparison. Consequently, the issue of the validity range of CFD codes for 3-D NRS applications has first to be addressed before the use of CFD may be considered as routine and trustworthy, as it is, for example, in the turbo-machinery, automobile and aerospace industries. Assessment of the reliability of CFD methodology in NRS applications represented the primary focus of the WG2 group.

[#] Working Group on the Analysis and Management of Accidents

Objectives and Scope

The main tasks of WG2 were originally defined as follows:

- Extend and consolidate the existing provisional WG2 document to the level of a CSNI report, to act as a platform for launching a web-based assessment database;
- Monitor and assess the current status of CFD validation exercises relevant to NRS issues;
- Identify gaps in the technology base and assess the prospect of them being closed in the near future;
- Identify experiments the data from which could be used as a basis for CFD benchmarking activities;
- Organise, as a spin-off activity, a series of international workshops to promote availability and distribution of experimental data suitable for NRS validation.

The group concentrated on single-phase phenomena, considering that two-phase CFD is not yet of sufficient maturity for a useful assessment basis to be constructed, and that identification of the areas which need to be developed (the task of WG3) should be undertaken first. Nonetheless, for completeness, those phenomena requiring multi-phase CFD have been identified in this document, but not elaborated upon. Where appropriate, reference is given to the WG3 document (NEA/CSNI/R(2010)2), where such issues are taken up and discussed in detail.

It was recognised that the nuclear community was not the primary driving force for the emergence of commercial CFD software during the early years of its development (1980s and 1990s), but could benefit nonetheless from the validation procedures undertaken in those industrial areas for which the basic thermal-hydraulic phenomena were similar. Consequently, it was necessary for the group to take full account of CFD assessment activities taking place outside the nuclear industry, and the present document reflects this wider perspective.

Organisation of the Document

The writing group met on average twice per year during the period March 2005 to May 2007, and coordinated activities strongly with the sister groups WG1 (Best Practice Guidelines) and WG3 (Multiphase Extensions). The resulting document prepared at the end of this time still represents the core of the present revised version, though updates have been made as new material has become available. After some introductory remarks, Chapter 3 lists twenty-three (23) NRS issues for which it is considered that the application of CFD would bring real benefits in terms of better predictive capability, and ultimately enhanced safety awareness in quantitative terms. This classification is followed by a short description of each specific safety issue, a highly condensed state-of-the-art summary of what has been attempted to date, what is still needed to be done to improve reliability, and a list of topical references.

Chapter 4 details the general assessment bases that have already been established, and discusses the usefulness and relevance of the work to NRS applications, where appropriate. This information is augmented in Chapter 5 by descriptions of the existing CFD assessment bases that have been established around specific NRS issues. Typical examples are experiments devoted to boron dilution, pressurised thermal shock, and thermal fatigue in pipes. The technology gaps which need to be closed to make CFD a more trustworthy analytical tool are listed in Chapter 6. Some deficiencies originally identified, such as limitations in the range of application of turbulence modelling, coupling of CFD with neutronics and system codes, and computer power limitations, have subsequently been filled, or partially filled. Most CFD codes currently being used in NRS applications have their own, custom-built assessment bases, the data being provided from both within and outside the nuclear community. These efforts are also documented.

Chapter 7 has been completely revised, since the CFD4NRS Workshop in Garching, Germany in 2006 has been followed by three more workshops in the series: XCFD4NRS (Grenoble, France, 2008), CFD4NRS-3 (Washington DC, USA, 2010) and CFD4NRS-4 (Daejeon, S. Korea, 2012). In addition, two OECD-sponsored CFD benchmark exercises have been organised by the CFD group within WGAMA, featuring topical issues of nuclear safety: thermal fatigue in T-junctions and turbulence generated downstream of a spacer grid in a rod bundle. Summary details are given.

Major Revisions

Several important additions to the original document have been made as a consequence of the later initiative within WGAMA to create a *CFD Task Group* to oversee the updating of the three Writing Group documents, and transfer the information to a Wiki environment on the NEA website. The updates and additions to the original WG2 document have been incorporated into this revised version. For easy reference, the modified sections are listed here.

- Section 3.15 (Induced Break) has been re-written in the light of more recent developments.
- Section 3.16 (Thermal Fatigue) has also been reworked, and extra references added.
- Section 3.25 (Sump Strainer Clogging) is a completely new addition to the document, making good an obvious earlier omission. Available validation data from the tests in Germany appear under Section 5.5.
- Section 5.3, which details the available assessment bases in the area of thermal fatigue, has been expanded to include the recent release of information on the issue deriving from operation of the sodium-cooled Phénix reactor, the tests from the WATLON series in Japan, and the recent OECD-Vattenfall CFD International Benchmark. The reference list has also been extended.
- Section 5.5 (Sump Strainer Clogging) is a new addition to the document, detailing the tests made at HZDR in Germany on the issue. A comprehensive reference list has also been added.
- Section 6.12 (Scaling and Uncertainty) represents a major overhaul of the material contained in the original document (which was compiled principally from documentation written in the context of the EC 5th FWP ECORA). The new material is very extensive, and includes sub-sections on the basis scaling issue, the various scaling methodologies in current use, an illustrative example relevant to CFD, the existing methods of uncertainty analysis in CFD, recommendations on new paths to follow, and a comprehensive reference list.
- Section 7 has been extended to include included information on the creation of a web portal to provide online access to the material contained in the Writing Group reports
- Annex 1 has been updated substantially to include details of the four CFD4NRS Workshops held to date, including the list of technical sessions and the conclusions and recommendations coming from the panel session debates.

Follow-Up Activities

During the time the Writing Groups were still meeting regularly, there was already discussion among the groups of how better to make use of the material collected. These thoughts manifested themselves in a proposal to WGAMA to extend and broaden the work beyond just the production of the three archival documents. The following ideas were put forward:

- Organise a new series of international workshops to provide a forum for experimenters and numerical analysts to exchange information;
- Establish a Wiki-type web portal to give online access to the information collected and documented by the Writing Groups, and provide a means for updating and extending the information by inviting reader participation; and

- Encourage nuclear departments at universities and research organisations to release previously restricted test data by initiating a series of international benchmarking exercises.

The CFD4NRS Workshops

The first of the workshops, which are all specifically focused on the application of CFD to nuclear reactor safety (NRS) issues, took place in 2006 under the acronym CFD4NRS, sponsored jointly by the OECD/NEA and the IAEA. There were 79 attendees. Papers describing CFD simulations were accepted only if there was a strong validation component. In total, 39 technical and 5 invited papers were presented. Most related to the NRS issues highlighted in this document, such as pressurised thermal shock, boron dilution, hydrogen distribution, induced breaks and thermal striping. Selected papers appeared in a special issue of Nuclear Engineering and Design (NED). The second workshop in the series, XCFD4NRS, took place in Grenoble, France in September 2008. Here, the emphasis was more on new experimental techniques and two-phase CFD. The workshop attracted 147 participants. There were 5 invited speakers, 3 keynote talks, 44 technical papers and 15 posters. Again, selected papers were collected in a special issue of NED. The third workshop, CFD4NRS-3, was held in Washington DC in September 2010 and its proceedings appeared during 2011 with selected papers in a topical issue of Nuclear Engineering and Design in 2012. The fourth workshop, hosted by KAERI, took place in Daejeon, Rep. of Korea in September 2012 with the proceedings published in early 2014 (http://www.oecd-nea.org/nsd/docs/2014/csni-r2014-4.pdf). The fifth workshop, CFD4NRS-5, was hosted by ETH Zurich in September 2014; at the time of writing, proceedings are being prepared and some papers have been selected for a special issue of Nuclear Engineering and Design. More details are given in Appendix 1.

Moving the Writing Group Documents to the Web

The activities of the three OECD/NEA Writing Groups on CFD were concluded at the end of 2007 with the completion, or near completion, of their respective CSNI reports. It was recognised, like any state-of-the-art report, these documents would only be up-to-date at the time of writing, and, given the rapidly expanding use of CFD in the nuclear technology field, the information they contained would soon become outdated, though perhaps less so for the WG1 document dealing with BPGs. To preserve their topicality, improvements and extensions to the documents were already 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. In a first step, the WG2 document in the form in which it appears as an archival document was uploaded to provide on-line access. The WG1 document has also since been uploaded, and the webpages for the WG3 document are under construction. Some details are given in Annex 2.

CFD Blind Benchmark Exercises

At a meeting of the chairmen of the NEA CFD Writing Groups in 2008, it was decided to utilize the organization within the *Special CFD Group* of WGAMA to launch the first of a series of international benchmark exercises. Both single-phase and two-phase flow options were considered. It was generally agreed that it would be desirable to have the opportunity of setting up a blind benchmarking activity, in which participants would not have access to measured data, apart from what was necessary to define initial and boundary conditions for the numerical simulation, until they had submitted their numerical predictions for evaluation. This would entail finding a completed, or nearly 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, and for the subsequent synthesis of the results.

Two such benchmarking exercises have since been conducted, and a third is at the planning stage. The first examined the issue of high-cycle thermal fatigue in a T-junction geometry, and was based on previously unreleased test data from a very careful experiment carried out at the Älvkarleby Laboratory of Vattenfall Research and Development in Sweden in November 2008. The benchmark activity ran from May 2009 (Kick-Off Meeting) to December 2010 (CSNI approval of the final report). In total, 29 participants submitted blind numerical predictions for synthesis. The second benchmark exercise focused on the ability of CFD codes to predict turbulence characteristics downstream of a spacer grid in a rod-bundle geometry. Special tests were carried out in the MATiS-H cold-flow facility at the Korea Atomic Energy Research Institute (KAERI) in early Spring 2012. Two spacer grids (of generic design), of the split type and swirl-type, were featured in the study. Computer Aided Design (CAD) files of the spacer grids were made available by KAERI to aid CFD mesh generation. The benchmark was launched in April 2011, and 25 blind numerical predictions collected one year later. The final benchmark report was approved by the CSNI in December 2012. Annex 3 gives more details of both the benchmark activities.

Results and recommendations

The use of CFD in many branches of engineering is widespread and growing, due largely to the considerable advancements made in software and hardware technology. With the advent of multi-processor machines, application areas are expected to broaden, and expectations on the potential benefits in employing CFD methodologies to increase. Accompanying this drive forwards is a need to establish quality and trust in the predictive capabilities of CFD codes, and, as a consequence of open public awareness, this message is particularly relevant to the application of CFD to nuclear reactor safety. There is a need therefore to quantify the trustworthiness of the CFD results obtained from NRS applications. The mandate of the CFD Writing Group on assessment, WG2, was to specifically address this issue. The earlier document (issued in January 2008) represented, at the time of writing, a compendium of the then current application areas. It provided a catalogue of experimental validation data relevant to these applications, identified where the gaps in information lie, and made recommendations on what should be done to fill them. Primary focus was given to single-phase flow situations.

A list of NRS problems for which CFD analysis is required, or is expected to result in positive benefits, has been compiled, and reviewed critically. The list includes safety issues of relevance to core, primary-circuit and containment behaviour, under both normal and abnormal operating conditions, and during accident sequences, as comprehensively as could be assembled with the resources available. The list may be taken to represent the current application areas for single-phase CFD in NRS, and to serve as a basis for assembling the relevant assessment matrices. Since CFD is already an established technology outside the nuclear technology area, suitable validation data from all available sources has been included in the document. It was found that the databases were principally of two types: those concerned with general aspects of trustworthiness of code predictions (e.g., ERCOFTAC, QNET-CFD, FLOWNET), and those focused on particular application areas (e.g., MARNET, NPARC, AIAA). It was concluded that application of CFD to NRS problems can benefit indirectly from these databases, and the continuing efforts to extend them, but that a comprehensive NRS-specific database would always be needed to complement them. Consequently, the established assessment databases relating to specific NRS issues has been catalogued separately, and more comprehensively discussed in the document. Areas here include boron dilution, flow in complex geometries, pressurised thermal shock and thermal fatigue, all of which have already been the subject of CFD benchmarking activities.

Also identified, from a modelling viewpoint, are the gaps in the existing assessment databases. For single-phase CFD applications, these devolve around the traditional limitations of computing power, controlling numerical diffusion, the appropriateness of the established turbulence models, and coupling to system, neutronics and (to a lesser extent) structure mechanics codes. There is also the issue of isolating the CFD problem. An example is the specification of initial conditions if only an intermediate part of a given reactor transient is to be simulated, a part in which 3-D flow phenomena are expected to be important.

Important new information is provided by the material presented at the series of CFD4NRS Workshops, four of which have taken place between 2006 and 2012. Here, numerical simulations with a strong emphasis on validation were particularly encouraged, together with the reporting of experiments which have provided high-quality data suitable for CFD validation. In addition, an important new contribution to the assessment database is the organisation of CFD benchmarking activities, also promoted by WGAMA. Two benchmarking exercises have so far been completed (in the area of thermal fatigue in a T-junction and turbulence generation downstream of a spacer grid in a rod bundle), and a third benchmark is being planned, based on a new experiment to be performed in the PANDA facility at PSI.

The present document thus represents an important milestone in establishing a comprehensive assessment database for the application of CFD to NRS problems. A second stage will involve updating the new information to the Wiki website to enable ready access to the information, and give encouragement for users to supply new information. CFD remains a very dynamic technology, and with its increasing use within nuclear safety there will be ever greater demands to document current capabilities, and prove trustworthiness by means of validation exercises. It is therefore anticipated that any existing assessment database will soon need to be extended. To prevent important information assembled from becoming obsolete, the following recommendations were made in the original WG2 document, and subsequently acted upon.

- *Set up and maintain a web-based centre to consolidate, update and extend the information contained in the document.* The webpages are now active on the NEA website, and the new information contained in this document will be uploaded to it in due course.
- *Provide a forum for numerical analysts and experimentalists to exchange information in the field of NRS-related activities relevant to CFD validation by holding further workshops in the CFD4NRS series, to provide information for building into the web-based assessment matrix.* Four such workshops have now taken place, and a fifth is planned for 2014.
- *Form a small task unit comprising one representative from each of the three Writing Groups, together with the NEA webmaster and secretariat, to act as the central organising body for the tasks here stated.* The task unit was formed, and became the central organising body for the CFD4NRS workshops and related benchmarking exercises.

In the longer term, new benchmarking exercises will need to be considered, based on suitable data already identified within this document, or on new data being presented at future workshops in the CFD4NRS series. It is not anticipated that these would be on the scale of an ISP, but would be of maximum two years duration from initial announcement to summary document. The reduced overhead will enable the benchmark organisers to respond quickly to changing directions in the application of CFD to nuclear reactor safety issues, and keep pace with the CFD4NRS workshop format, enabling the close links between them to be maintained.

TABLE OF CONTENTS

1. INTRODUCTION/BACKGROUND

Computational methods have supplemented scaled model experiments, and even prototypic tests, in the safety analysis of reactor systems for more than 35 years. During this time, very reliable system codes, such as RELAP-5, TRACE, CATHARE and ATHLET, have been formulated for analysis of primary circuit transients. Similar programs (such as SCDAP, MELCOR, GOTHIC, TONUS, ASTEC, MAAP, ICARE, COCOSYS/CPA) have also been written for containment and severe accident analyses.

The application of Computational Fluid Dynamics (CFD) methods to problems relating to Nuclear Reactor Safety (NRS) is less well developed, but is accelerating. The need arises, for example, because many traditional reactor system and containment codes are modelled as networks of 1-D or 0-D volumes. It is evident, however, that the flow in components such as the upper and lower plena, downcomer and core of a reactor vessel is 3-D. 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 misleading, producing erroneous conclusions.

One of the reasons why the application of CFD methods in Nuclear Reactor Safety (NRS) has been slow to establish itself is that transient, two-phase events associated with accident analyses are extremely complex. Traditional approaches using system codes have been successful because a very large database of phasic exchange and wall heat transfer correlations has been built into them. The correlations have been formulated from essentially 1-D special-effects experiments, and their range of validity well scrutinised. Data on the exchange of mass, momentum and energy between phases for 3-D flows is very sparse in comparison. Thus, although 1-D formulations may restrict the use of system codes in simulations in which there is complex geometry, the physical models are well-established and reliable, provided they are used within their specified ranges of validity. The trend has therefore been to continue with such approaches, and live within their geometrical limitations.

For containment issues, lumped-parameter codes, such as COCOSYS or TONUS-0D, include models for system components, such as recombiners, sprays, sumps, etc., which enable realistic simulations of accident scenarios to be undertaken without excessive computational costs. To take into account such systems in a multi-dimensional (CFD) simulation remains a challenging task, and attempts to do this have only recently begun, and these in dedicated ‘CFD-type’ codes such as GOTHIC, GASFLOW or TONUS-3D rather than with general-purpose CFD software.

The issue of the validity range of CFD codes for NRS applications has also to be addressed, and may explain why the application of CFD methods is not straightforward. In many cases, even for single-phase problems, nuclear thermal-hydraulic flows may lie outside the range of standard models and methods, especially in the case of long, evolving transient flows with strong heat transfer, and feed-back effects on system behaviour and neutronics.

It appears then that there exists a duality between system codes, with limited geometric capabilities and non-guaranteed control of numerical errors, but with sophisticated and highly trustworthy physical models, and which often run in real time for real reactor transients, and CFD, for which geometric complexity is no real issue, with modern numerical schemes, but for which, at least for two-phase and containment applications, the physical models require considerable further development, and for which massive parallel machine architecture is often required for real reactor applications.

The present activity arises from the need to critically assess[#] where CFD methods may be used effectively in problems relating to Nuclear Reactor Safety (NRS), and to demonstrate that utilisation of such advanced numerical methods, with large computer overheads, is justified, because the use of simpler engineering tools or 1-D codes have proven to be limited, or even inadequate.

From a regulatory perspective, a common approach to dealing with practical licensing issues is to use such simplified modelling, coupled with conservatism to cover the unknown factors. In this way, sufficient safety margins can be ensured. The advantage of the simplified modelling approach is that a large number of sensitivity studies can be 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 practise on a firm mathematical basis. However, a key issue is then to determine the degree of conservatism needed to cover the lack of physics embodied in the simplified models. Information can be obtained from mock-up experiments, but considerable care is necessary in extrapolating results to full scale. Moreover, the experiments themselves contain simplifications, and judging the conservatism involved in introducing the simplifications is itself quite difficult. The only way to ultimately ensure conservatism is to increase the margins, but this often places unwelcome constraints on plant effectiveness.

The trend is to gradually replace conservatism by a best-estimate methodology, coupled with an uncertainty evaluation. This process has already taken place in the context of system analysis codes 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 the multi-dimensionality of some of the safety analyses which need to be performed, always with the aim of reducing the conservatism associated with using simplified or inappropriate analysis tools. To gain acceptance in the licensing world, however, such investigations need to be underpinned by a comprehensive validation programme to demonstrate the capability of the technology to produce reliable results. Many examples are given in this document of how such reliability in the use of CFD can be achieved, where the limitations are, and what needs to be done to improve the situation. For single-phase applications, CFD is mature enough to complement existing analysis tools currently employed by regulatory authorities, and 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 appears unacceptable from a regulatory standpoint. No doubt, a solution will be found in due course.

The document is organised as follows. The objectives of the activity, which have been updated slightly from those originally set out in the CAPS (GAMA 2002 7, Revision 0, October 2002), are summarised in Chapter 2. The main body of the document begins with Chapter 3, which provides a list of NRS problems for which the need for CFD analysis has been recognised, and in most cases also actively pursued. A few references to each topic are provided for orientation purposes, but are not intended to be comprehensive. Two-phase problems requiring CFD are also listed for completeness, but all details are deferred to the companion WG3 document. Brief summaries of existing assessment databases (both from the nuclear and non-nuclear areas) are given in Chapter 4, and extended in Chapter 5 to include those databases centred around specific NRS issues. Here, the reference list is more comprehensive. From this information, the gaps in the assessment bases, with particular emphasis on NRS applications, are summarised in Chapter 6.

[#] The word *assess*, as used here, is a synonym for appraise, evaluate or judge.

A synthesis of the information gained from the papers presented at the series of CFD4NRS International Workshops is introduced in the first part of Chapter 7, with more complete details of the background material, scope and objectives, the presentations and poster sessions, and conclusions and recommendatons given in Annex 1. The Chapter also contains some suggestions for possible future CFD benchmarks for the primary circuit, core and containment, as compiled for the original release of this document. However, the subsequent sections of Chapter 7 describe the actual benchmark exercises actually carried out within the OECD/NEA initiative. Overall conclusions, recommendations and perspectives are provided in Chapter 8. Finally, Annex 1 gives details of the workshop programmes of the four CFD4NRS conferences held to date, including the summaries and recommendations made by participants on each occasion. Annex 2 contains a brief description of the web-based WG2 document, Annex 3 describes the two blind CFD benchmarks carried out to date, and Annex 4 contains a glossary of the acronyms used in the document.

2. OBJECTIVES OF THE WORK

The basic objective of the present activity is to provide documented evidence of the need to perform CFD simulations in NRS (concentrating on single-phase applications), and to assess the competence of the present generation of CFD codes to perform these simulations reliably. The fulfilling of this objective will involve multiple tasks, as evidenced by the titles of the succeeding chapters, but, in summary, the following items list the specifics:

- To provide a classification of NRS problems requiring CFD analysis
- To identify and catalogue existing CFD assessment bases
- To identify shortcomings in CFD approaches
- To put into place a means for extending the CFD assessment database, with an emphasis on NRS applications.

3. NRS PROBLEMS WHERE (SINGLE-PHASE) CFD ANALYSIS BRINGS REAL BENEFITS

Introduction

The focus here will be on the use of CFD techniques for single-phase problems relating to NRS. This is the traditional environment for most non-NRS CFD applications, and the one which has a firm basis in the commercial CFD area. NRS applications involving two-phase phenomena will be listed in this document for completeness, but full details are reserved for the WG3 document (Extension of CFD Codes to Two-Phase Flow Nuclear Reactor Safety Problems, NEA/CSNI/R(2007)15, in preparation), which addresses the extensions necessary for CFD to handle such problems.

The classification of problems identified by the Group is summarised in Table 1, and then, under appropriate sub-headings, a short description of each issue is given, why CFD especially is needed to address it, what has been achieved, and what further progress needs to be made. There are also moves within the nuclear community to interface CFD codes with traditional system codes. Identification of the needs of this combined approach is also contained in Table 1, and then addressed more fully in the subsequent sub-sections.

With some overlaps, the entries are roughly grouped into problems concerning the reactor core, primary circuit and containment, consecutively.

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 experiments, etc.)	Primary circuit	Operational	Single

	NRS problem	**System classification**	**Incident classification**	**Single- or multi-phase**
10	BWR/ABWR lower plenum flow	Primary circuit	Operational	Single/Multi
11	Water-hammer 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
23	Sump strainer clogging	Containment	DBA	Single/Multi

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

3.1 Erosion, Corrosion and Deposition

Relevance of the phenomena as far as NRS is concerned

Corrosion of material surfaces may have an adverse effect on heat transfer, and oxide deposits may accrue in sensitive areas. Erosion of structural surfaces can lead to degradation in the material strength of the structures.

What the issue is?

The secondary circuit of a Pressurised Water Reactor (PWR) is essentially made of carbon steel and copper alloys. Corrosion produces oxides, which are transported to the Steam Generators (SGs) and give rise to deposits (e.g., on the tube support plate). There are two effects due to the presence of sludge in the SGs:

- effect on the efficiency of the SGs;
- corrosion of SGs (plate and tube degradation).

In the primary circuit, the chemistry is different, but corrosion phenomena are also encountered, particularly on the fuel claddings.

The oxide layers resulting from corrosion have altered properties compared to the initial construction material. If the layers are thin enough, the effect on the overall structural integrity is negligible. Such a thin oxide layer is in fact protecting the structural material from further degradation. However, in certain circumstances, the oxide layer may be eroded, due to a local increase of wall shear stress. This is typically occurring at places where there is a sudden change of flow direction, for example at a channel entrance or sudden area change. In such circumstances, the protective oxide layer may be continuously eroded, leading to substantial changes in structure integrity.

What the difficulty is and why CFD is needed?

The prediction of the occurrence of such phenomena requires simulation at very small scales. It is important to understand and predict primary and secondary circuit corrosion occurrence as well as sludge deposition in order to control and limit their occurrence. System codes and component codes, which use either homogenisation or sub-channel analysis, cannot predict the highly localised phenomena associated with corrosion and deposition, and there is a need for a detailed flow field analysis, with focus on the wall shear stress prediction. (In the case of two-phase flow, it may require CFD extension to properly treat the two-phase boundary layer.) The rate of the erosion primarily depends on water chemistry (pH level, fluid oxygen content) and material properties, but it is also influenced by the following fluid-mechanics parameters:

- fluid local velocity;
- fluid local temperature;
- flow local quality.

These local parameters are geometry-dependent, and can only be predicted with a proper CFD model.

What has been attempted and achieved/what needs to be done (recommendations)?

Some successful applications of CFD in predicting erosion/corrosion already exist; e.g. Ref, 2. However, more work is needed to resolve near-wall mass and momentum transfer.

Proper modelling of erosion/corrosion requires investigation of both mass transfer and fluid flow in wall boundary layers. For that purpose, it is necessary to fully resolve the mass transfer boundary layer, which is typically an order of magnitude smaller than the viscous sub-layer. As a result, extremely fine grids in near-wall regions are required.

Further development of single-phase CFD models is required in the following areas:

- Investigation of the turbulent Schmidt number in near wall regions using: e.g. DNS approach
- Development of turbulence models in near wall regions, tailored for mass transfer predictions
- Development of erosion models
- Modelling of complex 3D geometries.

In Ferng et al. (2006), a methodology is presented to predict the wall thinning locations on the shell wall of feed water heaters. The commercial CFD code ANSYS-CFX 4.2 with an impingement erosion model implemented into an Eulerian/Lagrangian model of flow of steam continuum and water droplets enabled prediction of wear sites on the shell wall. These corresponded well with the measured ones obtained from a PWR located in the southern region of Taiwan. Droplet kinetic energy was used as an appropriate indicator of possible locations of severe wall thinning.

Ref. 1: Burstein G.T., Sasaki K., “Effect of impact angle on the erosion-corrosion of 304L stainless steel,” WEAR, 186-187, 80-94 (1995)

Ref. 2: A. Keaton, S. Nesic, “Prediction of two-phase erosion-corrosion in bends”, 2nd Int. Conf. CFD in the Minerals and Process Industries, CSIRO, Melbourne, Australia, 6-8 Dec. 1999.

Ref. 3: G. Cragnolino, C. Czaijkowski, W. J. Shack, NUREG/CR-5156, Review of Erosion-Corrosion in Single-Phase Flows, April 1988.

Ref. 4: McLaury, B.S., Shirazi S.A., Shadley I.R., Rybicki E.F., “Parameters affecting the accelerated erosion and erosion-corrosion”, Paper 120, CORROSION99, NACE International, Houston, TX (1999).

Ref. 5: Ferng, Y.M., Hsieh J.H., Horng, C. D. “Computational fluid dynamics predicting the distribution of thinning locations on the shell wall of feedwater heaters”, Nuclear Technology, 153, 197-207 (2006).

3.2 Core Instability in BWRs

This is a two-phase phenomenon, which is covered fully in the WG3 document.

Orientation

Flow instabilities in BWRs can induce power surges, because of the strong coupling between void fraction and neutronics. The coupling results in a feedback system that under particular conditions can be unstable. In these conditions, the core experiences neutron power surges, with a frequency of the order of 0.5 Hz, eventually leading to a reactor scram.

The prediction of local or out-of-phase oscillations requires detailed 3D calculations, both for the kinetics and thermohydraulic parts. A very detailed representation of the core and of its surroundings is desirable in order to obtain more reliable predictions. This includes a detailed nodalisation of the lower and upper plena and recirculation flow path.

Many computer codes have been used to predict stability behaviour in a BWR, but most of the available codes are based on drift-flux formulations. It is desirable to assess the benefits that could be achieved using two-fluid models for the prediction of channel stability. Moreover, a greater effort should be spent on benchmarking available codes against experimental data of real plant behaviour.

Ref. 1: Lahey and Moody, ISBN 0-89448-037-5, “The thermal-hydraulics of a boiling water nuclear reactor” ch.7.

Ref. 2: F. d’Auria et al., OCDE/GD(97)13, “State of the art report on BWR stability”.

Ref. 3: C.Demazière, I.Pázsit: “On the possibility of the space-dependence of the stability indicator (decay ratio) of a BWR”, *Ann.Nucl. Energy*, **32**, 1305-1322 (2005).

Ref. 4: J.Karlsson, I.Pászit: “Noise decomposition in Boiling Water Reactors with application to stability monitoring”, *Int J.of Nucl. Sci. and Eng.,* **128**, 225-242 (1998).

Ref. 5: D. Hennig: “A study on boiling water reactor stability behaviour”, *Nucl Technology,* **126**(1), 10-31 (1999).

Ref. 6: D. Ginestar et al., “Singular system analysis of the LPRM readings of a BWR in an unstable event”, *Int J of Nucl Energy Science and Technology* **2**(3), 253-265 (2006).

3.3 Transition boiling in BWRs – determination of MCPR

This is a two-phase phenomenon, which is covered fully in the WG3 document.

Orientation

BWRs TechSpec requires that during steady-state operation the MCPR (Minimum Critical Power Ratio) thermal limit is kept above the licensed safety value. The MCPR tends to be a limiting factor at high burnup conditions. The current trend to extend plant lifetime and increase the fuel cycle duration requires improvements to be made in the methods used in the licensing analysis to estimate this limit. The use of CFD codes could lead to a significant decrease in the present, conservative assumptions employed.

Ref. 1: Lahey and Moody, ISBN 0-89448-037-5, "The thermal-hydraulics of a boiling water nuclear reactor", ch. 4.

Ref. 2: General Electric Co., NEDO-10958, "GETAB – General Electric BWR Thermal Analysis Basis".

Ref. 3: Y.-Y. Hsu and R. W. Graham, Transport Processes in Boiling and Two-Phase Systems: Including Near-Critical Fluids, ANS, 1968, ISBN: 0-89448-030-8.

3.4 Recriticality in BWRs

This is a two-phase phenomenon, which is covered fully in the WG3 document.

Orientation

In a BWR severe accident, the first materials to melt are the control rods. This is due to the low melting temperature for the mixture of boron carbide and stainless steel. The situation can lead to core recriticality and runaway overheating transients. The resultant molten material accumulates on top of the lower support plate of the core. Some of it re-solidifies, supporting an accumulating melt pool. The supporting layer eventually breaks, and melt pours into the lower plenum.

Coolant penetration into the core during reflooding is assumed to occur due to a melt-coolant interaction in the lower plenum. No integral code is capable of describing all the necessary phenomena.

Ref. 1: NUREG/CR-5653, "Recriticality in a BWR Following a Core Damage Event," U.S. Nuclear Regulatory Commission, November 1990.

Ref. 2: W. Frid et al. "Severe accident recriticality analyses (SARA)", Nucl. Engrng. and Design, 209, 97–106 (2001).

3.5 Reflooding

This is a two-phase phenomenon, which is covered fully in the WG3 document.

Orientation

A large-break, loss-of-coolant-accident (LBLOCA) remains the classical design-basis-accident (DBA), in the sense that the emergency core-cooling (ECC) system has to be designed to be able to reflood the core and prevent overheating of the fuel cladding. During reflooding, multi-dimensional flow patterns occur. Though the physical phenomena are complex, CFD has the potential of following the details of the flow, with the aim of reducing uncertainties in current predictions made on the basis of 1-D system codes and 0-D lumped-parameter codes.

Ref. 1: R.T. Lahey, Jr. & F.J. Moody The Thermal-Hydraulics of a Boiling Water Nuclear Reactor, Second Edition, American Nuclear Society, La Grange Park, Il, 1993, ISBN 0-89448-037-5.

Ref. 2: F. D'Auria, F. De Pasquale, J. C. Micaelli, Advancement in the study of reflood phenomenology in typical situations of PWR plants, Proceedings of UIT (Unione Italiana di Termofuidodinamica) VII National Conference on Heat Transfer, 15-17 June 1989.

Ref. 3: A. Yamanouchi, Effect of core spray cooling in transient state after loss of coolant accident, Journal of Nuclear Science and Technology, 5,547–558 (1968).

Ref. 4: G. Yadigaroglu, R. Greif, K.P. Yu and L. Arrieta, Heat Transfer During the Reflooding Phase of the LOCA-State of the Art, EPRI 248-1, (1975).

3.6 Lower Plenum Debris Coolability and Melt Distribution

Relevance of the phenomenon as far as NRS is concerned

During a severe accident in a nuclear power plant, the integrity of the nuclear reactor core is lost, and it can relocate to the lower plenum and form a debris bed. If cooling of the debris bed is not sufficient to remove the generated decay heat, a melt-through of the reactor pressure vessel will occur.

What the issue is?

Estimates of debris coolability and melt relocation are highly empirical, and dependant on the particular design solutions used in the nuclear power plants. However, what is common to all the scenarios is the necessity to halt accident progression, remove the decay heat from the debris bed, and prevent melt-through of the vessel.

What the difficulty is and why CFD is needed?

The following key parameters have to be taken into account in proper modelling of cooling of a debris bed:

- flow driving force (gravitation, capillary forces);
- flow resistance for both laminar flow (small particle areas) and turbulent flow (large particle areas);
- dryout criteria;
- counter-current flow limitation (CCFL);
- multi-dimensional effects;
- transient behaviour.

What has been attempted and achieved/what needs to be done (recommendations)?

Current approaches remain empirical, and correlations are used to predict the heat transfer rate between particles and the cooling water. The water penetration through the bed is highly dependent on the bed structure (non-uniform particle distributions) and simplified approaches can be applied. CFD can be used to improve the accuracy of predictions in non-uniform beds. In particular, three-dimensional models of flow in a porous material will give better estimates of the water penetration rates, and relaminarisation due to different grain sizes.

Ref. 1: T.N. Dinh, V.A. Bui, R.R. Nourgaliev, J.A. Green, B.R. Sehgal, "Experimental and Analytical Study of Molten Jet Coolant Interactions: The Synthesis", *Int. J. Nuclear Engineering and Design*, 189, 299-327 (1999).

Ref. 2: T. G. Theofanous et al. "In-vessel coolability and retention of a core melt", *Nucl. Eng. Des*., 169, 1-48 (1997).

Ref. 3: Y. Maruyama, et al. "Experimental study on in-vessel debris coolability in ALPHA program", *Nucl. Eng. Des.*, 187, 241-254 (1999).

Ref. 4: D. L. Knudson et al. "Late-phase melt conditions affecting the potential for in-vessel retention in high power reactors", *Nucl. Eng. Des.*, 230, 133-150 (2004).

3.7 Boron Dilution

Relevance of the phenomenon as far as NRS is concerned

Boron concentration aims at controlling the power and subcriticality for shutdown conditions. Mechanisms C:\Program Files\Real\RealPlayer\DataCache\Login\index.html supposed to lead to boron diluted water are known (consequence of small break, SG leakage etc. (ee Ref. 1 for a review).

What the issue is?

The safety problem concerns the possible transport to the core of a diluted slug of water, and the related power excursion.

What the difficulty is and why CFD is needed?

The whole phenomenon modelling 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 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: (i) pump start-up, or (ii) natural circulation after water inventory restoration. Relevant part of the reactor for flow modelling concern at least the downcomer, the lower plenum, and possibly the pipework related to the transportation of the slug. CFD features of the simulation are the transient behaviour of the flow, the geometrical complexity of the computational domain, and the requirement of the precise mixing properties of the flow.

What has been attempted and achieved/what needs to be done (recommendations)?

Boron dilution has been considered within an International Standard Problem (ISP-43, based on a University of Maryland Thermalhydraulic Facility allowing the mixing of flows of different temperature within a reduced scale vessel model, see Ref. 2).

Another scaled (1/5th) model (ROCOM, Forschungszentrum Rossendorf) of the German PWR KONVOI has been considered for several test scenarios related to boron dilution transients (steady state, transient and cavity-driven flows may be considered). Some related results have been published (Ref. 1).

A third test facility is the Vattenfall model, built at Vattenfall Utveckling, Älvkarleby in 1992. It is a 1:5 scale model of the 3-loop Westinghouse PWR at Ringhals. The model has been used for several studies, including CFD simulations. International cooperation has been within the EUBORA project, and now the on-going FLOWMIX-R project, both of them EU 5th Framework programmes.

For these databases, successful CFD results have been claimed, and applications to existing reactors have also been reported.

A concerted action on Boron Dilution Experiments (EUBORA, 1998, 4th EC program) gathered several European countries involved in CFD applications for such problems. Many facilities provided relevant data: the EDF Bora Bora facility; the Rosendorf ROCOM facility; the UPTF facility; and the PSI Panda facility (see Ref. 5). The conclusion from the EUBORA project was that 3-D CFD does provide an effective tool for mixing calculations, though the code calculations, and the applied turbulent mixing models, have to be validated by experiments. The current status on assessment is deemed not to be complete, it was concluded. A large-scale test (scale 1:2 tentatively) was also suggested to provide confirmation data.

The ongoing EU-project FLOWMIX-R aims at describing relevant mixing phenomena in the PWR primary circuit. It includes a well-defined set of mixing experiments in several scaled facilities (Rossendorf, Vattenfall, Gidropress and Fortum) to provide data for CFD code validation. Calculations are performed for selected experiments using two commercial CFD codes (ANSYS-CFX, FLUENT). The applicability of various turbulence modelling techniques is being studied for both transient and steady-state flows. Best Practise Guidelines (BPGs) are being applied in these computations. Homepage for FLOWMIX-R is www.fz-rossendorf.de/FWS/FLOMIX.

Also, an OECD action has recently started concerning a coolant transient for the VVER-1000 (Ref. 3).

Questions regarding the relevance of a test facility, when compared to reactor functioning conditions, may concern: (i) Re numbers (lower for the test facility, see discussion in Ref. 4), and (ii) complexity of the lower plenum, which may be different and lead to different mixing properties. The first point is considered as non-crucial, the second one may depend on the reactor considered.

Ref. 1: T. Hoehne, H.-M. Prasser, U. Rohde, "Numerical coolant mixing in comparison with experiments at the ROCOM test facility", in proceedings of the ANS Conference, USA, 2001.

Ref. 2: T. Hoehne, "Numerical simulation of ISP-43 test using CFX-4", in proceedings of the ANS-ASME conference, Penn State University, 2002.

Ref. 3: NEA/NSC/DOC(2003) document on OECD/DOE/CEA VVER-1000 Coolant Transient Benchmark – 1st Workshop.

Ref. 4: T. Hoehne, "Coolant mixing in pressurized Power Reactor", 1999, in Proceedings of ICONE 7.

Ref. 5: H. Tuomisto, et al., "EUBORA - Concerted Action on Boron Dilution Experiments", FISA-99 Symposium on EU Research on Severe Accidents, Luxembourg, 29 November - 1 December, 1999.

Ref. 6: ISP-43: Rapid Boron Dilution Transient Experiment, Comparison Report, NEA/CSNI/R(2000)22.

Ref. 7: B. Hemström, R. Karlsson, M. Henriksson. "Experiments and Numerical Modelling of Rapid Boron Dilution Transients in a Westinghouse PWR". Annual Meeting on Nuclear Technology, Berlin, May 2003.

Ref. 8: T.S. Kwon, C.R. Choi, C.H. Song and W.P. Baek, "A three-dimensional CFD calculation for boron mixing behaviors at the core inlet", Proc. NURETH-10, Seoul (2003)

Ref. 9: C.R. Choi, T.S. Kwon and C.H. Song, "Numerical analysis and visulaization experimenet on behavior of borated water during MSLB aith RCP running mode in an advanced reactor", Nuclear engineering and design, (2007)

Ref. 10: H. Tinoco et al., "Physical modelling of a rapid boron dilution transient", Vattenfall Utveckling AB, Report VU-S93:B21, 1993.

3.8 Mixing, Stratification, Hot-Leg Heterogeneities

In-vessel mixing phenomena

Relevance of the phenomenon as far as NRS is concerned

PWRs have two to four coolant loops, depending on the design. It is important for reactor control that cold water fed from these loops is thoroughly mixed before entering the core otherwise the safe operation of the reactor could be compromised.

What the issue is?

The issue is the study of the mixing phenomena occurring in the downcomer and lower plenum of the reactor in the case of an accidental transient leading to asymmetric loop-flow conditions in terms of temperature or boron concentration. Transients such as Main Steam Line Break, accidental or inherent dilution transients are relevant to this issue. In these scenarios, flow in one or more of the hot legs is colder or non-borated with respect to the other loops. In the case of poor mixing, cold or low borated water can be injected into the core leading to recriticality returns, with a risk of cladding failure and fuel dispersion.

In general, the simulation of these transients requires the coupling of systems codes, to represent the whole primary circuit, and a part of the secondary circuit except the core. Core inlet conditions (flow rates, temperature or enthalpy) are deduced from vessel inlet conditions by the application of a mixing matrix. Up to now, the coupling is weak and mainly external (close-ups, boundary conditions, etc.), but attempts are being made to have a stronger coupling (see, for example, the OCDE/CSNI PWR Main Steam Line Break Benchmark).

Description of the difficulties and why CFD is needed to solve it

Mixing in the downcomer and lower plenum, up to now, as far as we know, have been modelled using mixing matrices obtained by extrapolation of steady-state test results, and not always with the actual lower plenum geometry (i.e. including downcomer and lower plenum internal structures), and not always under real operating conditions (in general, a constant mixing matrix is used). These matrices are then introduced as input to system codes, or used as an interface between a system code and a 3D core thermal-hydraulic code.

The use of CFD codes for the real reactor case, validated against data from the tests which have been used in defining the validation matrix, would represent a big step forward, since CFD offers the possibility to deal with the detailed geometry of the reactor and, in the “near” future, with transient flow conditions.

In the short term, CFD calculations would help identify the mixing laws used in the actual schemes (systems codes, coupled system, 3D core thermal-hydraulic and neutronics codes) in use, and in the medium term, one could imagine integration of a CFD code into the coupled chain: i.e. system, CFD, core 3D thermal-hydraulic and neutronics codes operating together. Finally, in the long term, if the capability of CFD codes is assessed for core thermal-hydraulic simulation, one could imagine the use of CFD for lower plenum and the core, coupled to 3D neutronics codes.

State of the art - recommendations

In a first step, one could focus on the application of CFD independent of any coupling with other types of codes. Up to now, CFD has been applied with some encouraging results for steady-state calculations of mixing phenomena in plena with internal structures (see, e.g., Hot Leg Heterogeneities, Section 3.8).

The mixing process of feedwater and reactor water in the downcomer of an internal-pump BWR (Forsmark 1 & 2) has been numerically modelled using the CFD code FLUENT/UNS. Earlier studies, with a very coarse model had shown that a new sparger design is necessary to achieve an effective HWC through improved mixing in the downcomer. This requires detailed and accurate modelling of the flow, not only for determining the mixing quality, but also for avoiding undesirable effects, such as increased thermal loading of internal parts.

A 90-degree sector model, as well as smaller sector models, was used. The 90-degree model covered one (of four) spargers, two main coolant pumps (of eight), and flow from the steam separators. Some results are presented in Ref. 2 below. No verification tests have so far been performed, but hydraulic model tests of 1:5 scale or larger have been suggested.

The main difficulty in the application of CFD codes to such problems are due to:

- the complexity and expanse of the geometry to be modelled: at least the four hot legs and junctions with the core vessel, the downcomer and the lower plenum, together with all their internal structures, resulting in a large number of meshes;
- the difficulty in building the mesh due to the quite different scales in the domain (from a few cms to several metres);
- the need to perform transient calculations, with or without coupling to system codes and 3D core physics codes.

Consequently, application of CFD codes in such a field requires, mainly:

- validated models, especially models of turbulence, to estimate the mixing in the lower plenum,
- good capacity to treat complex geometries of very different sized scales.

A second step will be to treat all the difficulties related to the coupling of CFD codes with system codes, other 3D component codes, and with 3D neutronics (see Section 5.2).

Ref. 1: OCDE/NEA – US/NRC PWR Main Steam-Line Break Benchmark, http://www.nea.fr/html/science/egrsltb/pwrmslbb/index.html

Ref. 2: Tinoco, H. and Einarsson, T., “Numerical Analysis of the Mixing and Recombination in the Downcomer of an Internal Pump BWR”, *Modelling and Design in Fluid-Flow Machinery*, 1997.

3.9 Hot Leg Heterogeneities

Relevance of the phenomenon as far as NRS is concerned

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.

Description of the issue

The issue refers to the estimation of the flow-rates in a PWR plant. Indeed, for safe running, the real primary flow rates in the loops and the core have to be checked to ensure they do not exceed the limiting design-basis values. The upper value is deduced from mechanical considerations regarding the assembly holding forces, and on the control rod falling time, the lower value is associated to the DNB risk protection signal.

The real primary flow rates are deduced from on-site periodic measurements.

For each loop, the flow-rate is determined from the following formula:

$$Q_{loop} = \frac{3.6 \times 10^{6}}{\rho_{CL}} \times \frac{W_{SG} - W_{RCP}}{H_{HL} - H_{CL}} \qquad /1/$$

with:

- W_{SG} : thermal power extracted from the SG, deduced from a heat balance on the SG secondary side,
- W_{RCP} : thermal power given by the Reactor Coolant Pump, obtained via the RCP power measurement,
- ρ_{CL} : water density, given by the water property determination,
- H_{HL} : Hot Leg enthalpy,
- H_{CL} : Cold Leg enthalpy.

These two enthalpies are deduced from temperature measurements of the Hot and Cold legs of the loop under consideration.

In order to check if the estimated value does not exceed the criterion, the uncertainty on the final value has to be estimated. This uncertainty is a combination of all the basic uncertainties resulting from the measurement devices, and to the methodology used to determine the different elements in Equation /1/.

By far the main source of uncertainty (about 10 times greater than the other sources) is related to the estimation of the hot-leg temperature. Two kinds of uncertainties are involved in this estimation:

- the first (easy to estimate) is generated by the measurement-chain precision;
- the second is due to a lack of representation of the three temperature measurement locations used to estimate the *average temperature* in regard to the *real average temperature*.

Concerning the second uncertainty, despite the mixing processes in the upper plenum, important temperature and flow heterogeneities are still present at the hot-leg instrumentation location, leading to uncertainties in the estimation of the real average temperature. Consequently, in order to quantify this error, the real average temperature of the hot-leg has to be estimated from specific experimental tests, from specific plant tests, and finally by calculation.

Description of the difficulties and why CFD is needed to solve them

Direct extrapolation of experimental results to the real plant is very difficult, and often leads to an overestimation of the uncertainty. The use of this overestimated value in the case of plant modifications (e.g., core loading, etc.) can give results which do not satisfy the safety criteria. Advanced methodologies based on CFD calculations are then required in order to reduce this overestimation.

State of the art - recommendations

The situation at present is that CFD calculations have shown encouraging results. They 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, composed of two rotating counter-current vortices. Nevertheless, some discrepancies remain, such as the location of the centre of these vortices along the hot-leg pipe.

The main difficulties in the application of CFD codes for such a physical issue are listed below.

- The complexity and the expanse of the geometry to be modelled – the upper part of the core, the upper plenum and the dome, with all their internal structures, and the hot leg – and the very different scales (from 1 cm to a metre) of all the structures, lead to very difficult meshing problems, and to very expensive computations (involving several millions of computational cells).
- There are complexities involved in specifying the boundary conditions (core outlets, inner flow-rates in the lead tubes,…), and difficulties in initialising the turbulence levels.
- Very fine representation of the turbulent phenomena is required to localise the vortices in the hot leg. Consequently, application of CFD codes in such a field requires validated models, especially models of turbulence, to estimate mixing in the upper plenum and vortex development in the hot leg.

A good capacity to treat complex geometries, of very different scales, is also required.

Ref. 1: Rohde, U.; Höhne, T.; Kliem, S.; Hemström, B.; Scheuerer, M.; Toppila, T.; Aszodi, A.; Boros, I.; Farkas, I.; Muehlbauer, P.; Vyskocil, V.; Klepac, J.; Remis, J.; Dury, T., Fluid mixing and flow distribution in the reactor circuit – Part 2: Computational fluid dynamics code validation, Nuclear Engineering and Design (2007)

Ref. 2: Kliem, S.; Kozmenkov, Y.; Höhne, T.; Rohde, U., Analyses of the V1000CT-1 benchmark with the DYN3D/ATHLET and DYN3D/RELAP coupled code systems including a coolant mixing model validated against CFD calculations, Progress in Nuclear Energy 48(2006), 830-848

Ref. 3: Höhne, 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, Nuclear Engineering and Design Volume 236(2006)Issue 12, 1309-1325

3.10 Heterogeneous Flow Distributions

Steam generator tube vibration (fluid/structure interaction)

Relevance of the phenomenon as far as NRS is concerned

Vibrations of the steam generator tubes are due to hydraulic forces arising from the flow around the tube bends; this is a fluid/structure interaction problem. The vibrations mainly concern the part of the generator where either cross-flows develop (as, for example, for the single-phase flow at the generator inlet) or two-phase flows take place (in the evaporation region). Excessive vibrations of the tubes can lead to tube rupture. If this occurs, there will be mixing of primary and secondary circuits, and a (nominal at least) breach of the primary containment barrier. Improved understanding of the phenomena can lead to improvements in geometry, and better inspection procedures.

What the issue is?

Flow-induced vibration is significant at the U-bend section of the tubes, and anti-vibration bars are installed in some designs to restrict the amplitude of the vibration. A global understanding of the vibration excitation mechanism is proposed in Ref. 1, as well as a collection of reference data. Actual vibration modelling relies on estimation of excitation sources, hydrodynamic mass, damping phenomena, mean velocity, void fraction, etc., without the support of CFD. However, a better (assessed) prediction of such quantities may come from a finer flow description, and knowledge of local, small-scale quantities.

What the difficulty is and why CFD is needed to solve it?

System codes, such as RELAP5, cannot model the flow-induced vibration, or the mechanical interaction between the fluid and the structure. The coupling of the fluid and structure calculations is generally difficult, since (at least for Lagrangian modelling approaches) the mesh structure for the fluid calculation may change due to the motion of the structure. The relevant description should provide realistic mean values for future vibration models, and local values for coupled fluid/structure modelling in regions of complex flow. Both single-phase and two-phase flows are involved. For the first, existing models may provide some details, even if suitable assessment is required. Two-phase flow solvers may not yet be considered mature enough to provide relevant information for such phenomena.

What has been attempted and achieved / What needs to be done (recommendations)?

Some new experiments are proposed in Ref. 1, to complement those being conducted by CEA: for example, the Panachet experiment, which considers single-phase cross-flow over a matrix of tube bundles. Also noteworthy are the first attempts at simulation using a CFD tool. Fluid-structure interaction is not taken into account in many commercial CFD codes, though developments are now underway (see Section 6.9). Coupling of a reliable two-phase CFD code, if one exists, and a computational structural dynamics code is necessary to calculate the U-tube vibration, since the structural motion has a feed-back on the flow dynamics.

Ref. 1: “Flow induced vibration: recent findings and open questions”, Pettigrew, Taylor, Fisher, Yetisir, Smith, Nuclear Engineering and Design, 185, 249-276 (1998).

Ref. 2: I-C. Chu and H.J. Chung, “Fluid-Elastic Instability of Straight Tube Bundles in Air-Water Two-Phase Cross-Flow,” Proceedings of ICAPP `05, Paper 5668, Seoul, Korea, May 15-19, 2005.

Ref. 3: H.J. Chung and I.-C. Chu, “Fluid-Elastic Instability of Rotated Square Tube Array in Air-Water Two-Phase Cross-Flow,” Nuclear Engineering and Technology, Vol. 38, pp. 69-80, 2006.

Ref. 4: I.-C. Chu, H.J. Chung, C.H. Lee, H.H. Byun, and M.Y. Kim, “Flow-Induced Vibration Responses of U-Tube Bundle in Air-Water Flow,” Proceedings of PVP2007, PVP2007-26777, July 22-26, 2007, San Antonio, Texas, USA.

Ref. 5: K. W. Ryu, B. H. CHo, C. Y. Park, S. K. Park, “Analysis of fluid-elastic instability for KSNP steam generator tube and its plugging effect at central region”, Proceedings of PVP2003, July 20-24, 2003, Cleveland, Ohio, USA.

3.11 BWR/ABWR Lower Plenum Flow

Relevance of the phenomenon as far as NRS is concerned

There are many pipes in the lower plenum of a BWR or ABWR reactor. Two phenomena are relevant to NRS. One is the stress induced by flow vibration, which may cause these pipes to 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.

What the issue is?

In an ABWR, the reactor internal pumps are newly installed at the side, near the base of the reactor pressure vessel. (Fig. 1, Section 3.22) The following two problems are to be solved.

(1) Many internal structures, such as guidance pipes of control rods and instrumentation pipes for neutron flux detection, are situated close together in the lower plenum. It is necessary to check the integrity of these structures against flow induced-vibration stresses (Fig.2, Section 3.22).

(2) In an ABWR, partial operation of the reactor internal pumps is accepted. However, it is necessary to check that the coolant is uniformly distributed to the reactor core during such operation.

What the difficulty is and why CFD is needed to solve it?

Many internal structures are located close together in the lower plenum. At a time of partial pump operation, inverse flow can occur in the leg attached to the pump which has stopped. CFD codes are effective in evaluating the flow field in such complicated situations.

What has been attempted/achieved so far and what needs to be done?

The three-dimensional flow field in the reactor vessel has been evaluated successfully using the CFD code STAR-CD, with the standard k-epsilon turbulent model.

Ref. 1: S. Takahashi, et al., "Evaluation of Flow Characteristics in the Lower Plenum of the ABWR by using CFD Analysis", ICONE-11, Tokyo, JAPAN, April 20-23, 2003.

Ref. 2: J.H. Jeong, B.S. Han, “A CFD analysis of coolant flow in a PWR lower plenum without geometrical simplification”, ICONE-13, Beijing, China, 2005.

Ref. 3: J.H. Jeong, J.P. Park, and B.S. Han, "Head Loss Coefficient Evaluation Based on CFD Analysis for PWR Downcomer and Lower Plenum", NTHAS5, Jeju, Korea, November 26- 29, 2006

3.12 Water-Hammer Condensation

Relevance of the phenomenon as far as NRS is concerned

Fast closing (or even opening) of valves induces strong pressure waves, which propagate through the circuit, both in the primary and secondary loops. The dynamic effects on the pipework could induce damage, and are therefore a safety concern.

What the issues are?

Water-hammer is most often investigated with respect to the mechanical loads applied to the pipe structure, resulting from pressure waves. This is connected to the study of ageing phenomena of nuclear pressure vessel materials.

What the difficulty is and why CFD is needed?

The main issue concerns the loads applied to the structure. This implies knowledge of additional quantities, such as condensation speed, velocity and pressure distributions, from which depends the mechanical loading to the pipes. All these phenomena are characterised by very fast transients. The simulation typically requires very small time steps, and may be conducted using a one-dimensional code. Three-dimensional codes are required when volume effects are involved, for example in the hot leg.

The water-hammer phenomenon can develop along with stratification (thermal or phase induced), and this also has three-dimensional features: occurrence of radial pressure distributions [1] and three-dimensional turbulence effects. Code assessment needs to take care of the different possible geometries: straight pipes, elbows, change of pipe diameter, etc. The accurate evaluation of these quantities may require CFD.

What has been attempted and achieved/What needs to be done (recommendations)

Basic considerations for code assessment may be required for waves developing in liquids and gases: examples are air and water [2], and subcooled water and steam for vertical and/or horizontal pipes [3]. Available measurements would concern pressure at different positions in the pipes, and, in particular, in sensitive areas, such as the measurement of the condensed phase at the end of the pipe.

Results of the WAHALoads (Two-Phase Flow Water Hammer Transients and Loads Induced on Materials and Structures of Nuclear Power Plants) EC programme may be of interest in the near future. The WAHALoads group may select and open for public use a set of relevant experiments undertaken during the program. This should be done in the spirit of a benchmarking activity and related code assessment.

Ref. 1: Gaddis and Harling, “Estimation of peak pressure-rise in a piping system due to the condensation induced waterhammer phenomenon”, Proceedings of ASME/JSME Fluid Engineering Division Summer Meeting, 1999.

Ref. 2: K. W. Brinckman, M. A. Chaiko, “Assessment of TRAC-BF1 for waterhammer calculations with entrapped air”, J. of Nuclear Technology, 133(1), 133-139 (2001).

Ref. 3: Giot, M., Prasser, H.M., Dudlik, A., Ezsol, G., Habip, M., Lemonnier, H., Tisej, I., Castrillo, F., Van Hove, W., Perezagua, R. & Potapov, S., “Twophase flow water hammer transients and induced loads on materials and structures of nuclear power plants (WAHALoads)” FISA-2001 EU Research in Reactor Safety, Luxembourg 12-15 November 2001, EUR 20281, 176-187, G. Van Goethem, A. Zurita, J. Martin Bermejo, P. Manolatos and H. Bischoff, Eds., EURATOM, 752p., 2002.

Ref. 4: Prasser, H.-M., Böttger, A., Zschau, J., Baranyai, G., and Ezsöl, Gy., "Thermal Effects During Condensation Induced Water Hammer Behind Fast Acting Valves In Pipelines", International Conference On Nuclear Engineering ICONE-11, 20-23 April, 2003, Shinjuku, Tokyo, Japan, Paper no. ICONE11-36310.

Ref. 5: Bogoi, A., Seynhaeve, J.M., Giot, M., “A two-component two-phase bubbly flow model - Simulations of choked flows and water hammer” 41th European Two-Phase Flow Group Meeting in Norway and 2nd European Multiphase Systems Institute Meeting, May 2003.

Ref. 6: Altstadt, E., Carl, H., Weiss, R., “Fluid-Structure Interaction Experiments at the Cold Water Hammer Test Facility (CWHTF) of Forschungszentrum Rossendorf”, Annual Meeting on Nuclear Technology, 2002, 14–16 May, 2002, Stuttgart, Germany.

3.13 Pressurised Thermal Shock (PTS)

Relevance of the phenomenon as far as NRS is concerned

PTS is related to the ageing of the vessel (because the mechanical resistance of the structure decreases with age). The events of concern are cold-water injections – which would, for example, accompanying a Loss of Coolant Accident followed by Emergency Core Cooling System (ECCS) injection; a Main Steam Line Break; a steam generator tube rupture; a small break loss of coolant; etc. (see Refs. 1 and 2) – that may lead to a thermal shock. Both single-phase and two-phase flow situations may occur.

What the issue is?

The issue is to predict the temperature (and the related thermal stresses) for the part of the vessel subjected to thermal shock, in order to investigate thermal fatigue, and the mechanical stresses to the vessel. Limited to the CFD concerns, the temperature of the vessel is determined through the temperature of the water in contact with the walls, and is influenced by turbulence, stratification (for both single- and two-phase situations), and, in the case of two-phase flows, by the condensation rate (the issue is connected with the direct-contact-condensation issue). The CFD issues are to take into account these features for the whole transient (which may last for several hundreds of seconds), for complex geometries (downcomer, upper plenum, and connected pipes), and for complex flow patterns (stratified flows, jets, plume development in the downcomer, etc.).

What the difficulty is and why CFD is needed?

The temperature of the vessel is determined through the temperature of the fluid in contact with it, and is influenced by turbulence (which enhances mixing), stratification (for both single- and two-phase situations), and by the condensation rate (for two-phase flow).

The whole phenomenon is unsteady, 3-D, and the precise determination of all the parameters is complex. The existing reported simulations concern single-phase flow, whereas simulations of two-phase flows in such situations are just beginning. Concerning single-phase flows, however, the precise description of the problem is reported to require turbulence models where both low Reynolds effects, laminar to turbulence transition and buoyancy effects need to be taken into account (Ref. 3).

What has been attempted and achieved/what needs to be done (recommendations)?

No systematic assessment has yet been reported, and only the system codes may be considered as validated against this problem. Although the single-phase CFD applications seem mature enough to be used, reported attempts were not all successful (see Ref. 3), and the further use of relevant experimental data and turbulence modelling improvement has been suggested (see Ref. 5).

For CFD, two assessment methods may be considered. Firstly, an assessment has to be made of the ability of a method to reproduce a particular phenomenon within the whole transient: one may consider the capability of the method to solve unsteady, coupled problems between the structure and the flow (thermal fatigue issue), the ability to describe stratification, to estimate condensation for different flow patterns (reported uncertainties concern for example the Heat Transfer Coefficient (HTC) inside the plumes). Secondly, the assessment should take into account an entire thermal shock sequence with the complete geometry. Reported relevant experiments are:

COSI: the COSI experiment is scaled 1/100 for volume and power from a 900 MW PWR and allows various flow configurations. Simulations representing small break LOCA thermal-hydraulic conditions, and including temperature profiles at various axial positions in the pipe and condensation rates, are reported in Ref. 1, and validation of models on Separate-Effect tests are reported in Ref 7.

An international study concerning PTS (International Case RPV PTS ICAS) has been completed, and proposed comparative assessment studies for which CFD codes could be used (Ref. 4). Reported data used for thermal-hydraulic tests concern the Upper Plenum Test Facility (UPTF) in Manheim. Particular attention was paid to thermal-hydraulic mixing. A first description of UPTF facility is available at the following web-site: http://asa2.jrc.it/stresa_framatome_anp/specific/uptf/uptffac.htm, or at http://www.nea.fr/abs/html/csni1004.html.

For both single- and two-phase flows, model improvement seems to be required. (See also the requirements for two-phase flows models in the work of the writing group on two-phase flow CFD.)

Ref. 1: P. Coste, "An approach of multidimensional condensation modelling for ECC injection", in the Proceedings of the European Two Phase Flow Group Meeting, 2003.

Ref. 2: H.K. Joum, T.E. Jin, "Plant specific pressurized thermal shock evaluation for reactor pressure vessel of a Korean nuclear power plant", in the Proceedings of the International Conference on Nuclear Energy in Central Europe, 2000.

Ref. 3: J. Sievers, HG Sonnenburg, "Modelling of Thermal Hydraulic Loads and Mechanical Stresses on Reactor Pressure Vessel", presented at Eurosafe 1999.

Ref. 4: "Comparison report of RPV pressurized thermal shock international comparative assessment study (PTS ICAS)", 1999, NEA/CSNI/R(99)3 report.

Ref. 5: "Advanced Thermohydraulic and neutronics codes: current and future applications", 2001, NEA/CSNI/R(2001)1/VOL1 report.

Ref. 6: D. Lucas et al., "On the simulation of two-phase flow Pressurized Thermal Shock", Proc. 12th Int. Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-12) Pittsburgh, Pennsylvania, U.S.A., September 30-October 4, 2007.

Ref. 7: W. Yao, P. Coste, D. Bestion, M. Boucker, "Two-phase pressurized thermal shock investigations using a 3D two-fluid modelling of stratified flow with condensation", Proceedings of the NURETH-10, Seoul, Korea, 2003.

3.14 Pipe Break

Relevance of the phenomenon as far as NRS is concerned

Transient pressure forces occur on the structures following a large pipe break, and are of importance for various reactors. Inside the reactor vessel, the decompression waves will produce dynamic loadings on the surfaces of the vessel internals, such as the core shroud and core grids of a BWR.

What the issue is?

This issue is an important example of the need to predict accurately three-dimensional, transient pressure fields, in order to estimate dynamic loadings on the internals. Structural analysis nowadays has to include dynamic loads, even for loss-of-coolant accidents.

What the difficulty is and why CFD is needed?

The decompression process is a highly three-dimensional and transient phenomenon, so it is well suited for a 3D CFD simulation. During the first phase, before flashing of the reactor water begins, a single-phase CFD model could be used. After flashing has started, a two-phase model is necessary to describe the decompression process, since then two-phase effects are dominant.

What has been attempted and achieved/what needs to be done (recommendations)?

CFD analysis of a steam line break in a BWR plant was part of a qualifying programme before the replacement of core grids at Units 1 and 2 at Forsmark NPP, Sweden, [Ref. 1]. The study was based on the assumption that the time scale of the transient analysis is smaller than the relaxation time of the water-steam system.

The results displayed a rather complex behaviour of the decompression, and the instantaneous forces computed were approximately twice those estimated in the past using simpler methods. It was pointed out that, at longer times, a two-phase model is necessary to describe the decompression. The results have not been validated against experiments, however.

During the last few years, several other simulations of rapid pipe breaks have been performed for Swedish reactors, also with no possibilities to compare with experimental results. Validation against HDR Experiments was therefore foreseen. In the early 1980s, the HDR (Heissdampfreaktor) blow-down experiments had been performed in Karlsruhe, Germany [Refs. 2 and 3]. The HDR rig consists of a blow-down nozzle, and a large pressure vessel, including internals (core barrel). The blow-down experiment V31.1 has been used for validation of numerical simulations, first using system codes, such as RELAP [e.g. Ref. 4], and later also with CFD (or CFD-like) codes. Lars Andersson et al. [Ref. 5] has presented simulation results using Adina-FSI (a coupling between the codes Adina-F (CFD) and the Adina structure solver) at the ASME PVP 2002 conference. The conclusions were that the results 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. Without FSI, the simulations show a factor 2 higher frequency for the pressure oscillations, and the amplitudes were generally higher. The conclusion was that the effects of FSI have to be included to obtain reliable results.

Ref. 1: Tinoco, H., “Three-Dimensional Modelling of a Steam-Line Break in a Boiling Water Reactor”, Nuclear and Engineering, 140, 152-164 (2002).

Ref. 2: Wolf, L., “Experimental results of coupled fluid-structure interaction during blow down of the HDR-vessel and comparison with pre- and post-test prediction”, Nuclear Engineering and Design, 70, pp. 269-308 (1982).

Ref. 3: HDR Sicherheitsprogramm. Auswertung von Dehnungsmessungen am HDR-Kernmantel und vergleich mit Spannungsberechnungen bei Bruch einer Reaktorkühlmittelleitung. Auswertebericht Versuchsgruppe RDB-E II. Versuche: V31.2, V32, V33, V34.

Ref. 4: Müller, F. Romas, A., “Validation of RELAP-5 against HDR-experiments”, DNV-Kärnteknik, 2002.

Ref. 5: Andersson, L., Andersson, P., Lundwall, J., Sundqvist, J., Veber, P., “Numerical Simulation of the HDR Blowdown Experiment V31.1 at Karlsruhe”, PVP-Vol. 435, Thermal-Hydraulic Problems, Sloshing Phenomena and Extreme Loads on Structures, ASME 2002.

3.15 Induced Break

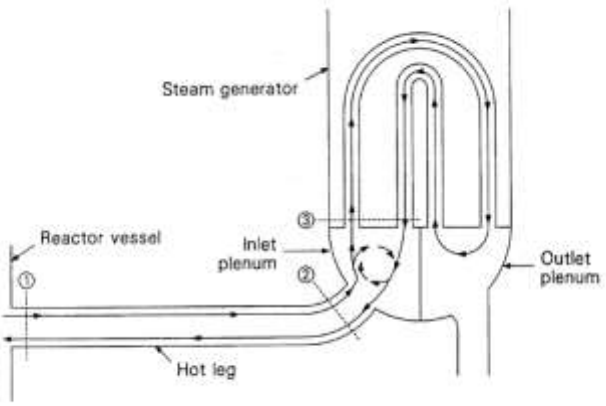
Relevance of the phenomenon as far as NRS is concerned

This scenario is of direct safety relevance because it involves the potential for a steam generator tube rupture during a severe accident scenario, which could lead to the release of fission products bypassing the containment.

Description of the issue

This subject is devoted to PWR induced break during a high pressure severe accident (e.g., due to total station blackout with a loss of secondary feed water). In this kind of scenario, the core is uncovered, heat is carried away from the fuel by steam in a process of natural circulation to structures in the reactor coolant system, including the upper vessel, hot leg, and steam generator tubes. The loop seals remain filled with water, and full primary loop circulation is blocked. A counter-current, natural circulation pattern in the hot leg and steam generator (with direct and reverse circulation in different SG tubes) ensues, as has been experimentally observed.

The temperatures during the severe accident ultimately lead to a thermally induced failure in the primary coolant loop. The flow field and heat transfer details determine whether the failure occurs within the vessel, in the reactor coolant piping system, or in the steam generator tubes, this providing a leak path that bypasses the containment. Details of the three-dimensional flow fields and heat transfer mechanisms are needed in order to predict the likely failure location.



The key parameters addressed in these evaluations are the magnitude of the natural circulation flows in the reactor coolant system piping and steam generator tube bundle, as well as the mixing and entrainment that occurs within the hot leg and steam generator inlet plenum.

Description of the difficulties and why CFD is needed to resolve them?

The thermal-hydraulic and core-degradation modelling of this severe accident scenario is generally performed using lumped- parameter codes such as SCDAP/RELAP5, CATHARE/ICARE, etc. The efficiency of the lumped-parameter approach makes it feasible to predict the transient behaviour of the entire reactor coolant system over extended periods of time. These codes, however, do not implicitly model the three-dimensional mixing and entrainment behaviour important for determining the magnitude of the natural circulation flows in the system. The system codes must rely on pre-determined flow paths and mixing ratios that are used to adjust the system code predictions to ensure consistency with experimental observations, or predictions from multi-dimensional tools such as CFD. CFD predictions have been used to extend the limited small-scale experimental database to a variety of full-scale conditions. Some of the key issues that have been studied using CFD predictions include the following:

- hot leg flow rate;
- steam generator tube bundle flow rate;
- tube bundle flow and temperature distributions;
- mixing and entrainment in the hot leg and steam generator inlet plenum;

- impact of the pressurizer surge line;
- impact of steam generator tube leakage on the natural circulation flows;
- impact of inlet plenum and loop geometry variations.

State of the art - recommendations

To date, CFD has been applied with some encouraging results for steady-state calculations of the reactor case [1-4], and for one experimental validation case [5]. The main difficulties in the application of CFD codes to such accident scenarios are listed here.

- The complexity and expanse of the geometry to be modelled: at least one hot leg with the pressuriser surge line, the primary side of the steam generator, including both plena (inlet and outlet), the SG tubes, and possibly the vessel upper plenum.
- The extent of this domain, especially the large number of steam generator tubes, presents a challenge to the CFD modeller. In addition to the large domain, the modeller is faced with complex, buoyancy-driven turbulent flows of steam and hydrogen, and the potential for radiative heat exchange between the structure and the optically-thick, high-pressure steam mixture.

Consequently, application of CFD codes in such a field requires:

- validated models, especially models of turbulence, to estimate mixing and stratification;
- a validated model of radiative heat exchange (with steam and hydrogen at high temperatures);
- simplified, but accurate, nodalisation of the tube bundle – the solutions one can imagine are to couple 1D and 3D models, or to define some equivalent (Ref. 4) to reduce the size of the mesh;
- validated models of the depressurisation induced by the opening of the safety valves (i.e. compressible or quasi-compressible model).

Ref. 1: H. Mutelle, U. Bieder "Study with the CFD Code TRIO_U of Natural Gas Convection for PWR Severe Accidents", NEA and IAEA Workshop: Use of computational fluid dynamics (CFD) codes for safety analysis of reactor systems including containment - PISA ,Italy, November 11-15, 2002.

Ref. 2: U. Bieder, C. Calvi, H. Mutelle "Detailed thermal hydraulic analysis of induced break severe accidents using the massively parallel CFD code TRIO_U/PRICELES", SNA 2003 International conference on super computing in nuclear applications, Paris, France, 22-24 Sept. 2003.

Ref. 3: C.F. Boyd, D.M. Helton, K. Hardesty, "CFD Analysis of Full-Scale Steam Generator Inlet Plenum Mixing During a PWR Severe Accident", NUREG-1788, May 2004.

Ref. 4: C.F. Boyd, K.W. Armstrong "Computational Fluid Dynamics Analysis of Natural Circulation Flows in a Pressurized-Water Reactor under Severe Accident Conditions," NUREG-1922, March 2010.

Ref. 5: C.F. Boyd, K. Hardesty "CFD Analysis of 1/7th Scale Steam Generator Inlet Plenum Mixing during a PWR Severe Accident", NUREG-1781, September 2003.

3.16 Thermal Fatigue in Stratified Flows

Relevance of the phenomenon as far as NRS is concerned

Thermal stratification, cycling and striping phenomena may occur in different piping systems of nuclear plants. They 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 and mix together.

What the issue is?

Often the phenomena are caused by defective valves through which hot (or cold) coolant leaks into cold (or hot) coolant. Damage due to thermal loadings has been reported in mixing tees of both the primary and secondary loops, for both sodium-cooled and water-cooled reactors. Static mixers have sometimes been inserted once first inspections have indicated cracks. Thus, in general, the more common thermal fatigue issues are understood, and can be controlled. However, some incidents indicate that certain information on the loading in the mixing zone, and its impact on the structure, is still missing.

In accident conditions, plume and stripe cooling in the downcomers of LWRs may occur. Different flow patterns are present, depending on the flow rates in the ECC injection nozzles, and the downcomer water levels. Two-phase flow may occur when cold water is heated through an isolation device by hot water, causing the cold water on the other side to rise above the saturation temperature. One may encounter stratified flows, low velocities, and sometimes the presence of air due to degassing. There might also be low-frequency flow fluctuations associated with temperature fluctuations, which may lead to thermal fatigue.

What the difficulty is and why CFD is needed to solve it?

CFD is able to predict the thermal loadings on the metallic structures. Single-phase CFD may need to include LES (Large Eddy Simulation) turbulence modelling to be able to predict the frequency and amplitude of the large-scale fluctuations, both of which are important parameters for the associated structural and failure analyses.

What has been attempted and achieved/what needs to be done (recommendations)?

Current studies are focussed on single-phase situations. Development of a two-phase CFD code able to handle stratified flows with temperature and density stratification, and with turbulent mixing effects, and possibly using LES for the liquid, flow would be useful for some two-phase situations.

Ref. 1: T. Muramatsu, "Numerical analysis of non-stationary thermal response characteristics for a fluid-structure interaction system", Journal of Pressure Vessel Technology, 121, 276, 1999.

Ref. 2: K.-J. Metzner, U. Wilke, "European THERFAT project – thermal fatigue evaluation of piping system Tee-connections", Nucl. Engng. Des., 235, 473-484 (2004).

Ref. 3: J. Westin et al., "Experiments and Unsteady CFD Calculations of Thermal Mixing in a T-Junction", Proc. Int. Workshop on Benchmarking of CFD Codes for Application to Nuclear Reactor Safety (CFD4NRS), Garching, Munich, Germany, 5-7 September 2006 (CD-ROM).

Ref. 4: K.C. Kim, M.H. Park, H.K. Youm, J.H. Kim, "Thermal Stratification Phenomeon in a Branch Pipping with In-Leakage", Proceedings of Nureth-10, 2003 (CD-ROM).

Ref. 5: K.C. Kim, M.H. Park, H.K. Youm, S.K. Lee, T.R. Kim and J.K. Yoon, "An Unsteady Analysis on Thermal Stratification in the SCS Piping Branched Off the RCS Piping", Proceedings of ASME PVP, 2003.

Ref. 6: H.K. Youm, K.C. Kim, M.H. Park, T.E. Jin, S.K. Lee, T.R. Kim and J.H. Kim, "Fatigue Effect of RCS Branch Line by Thermal Stratification", Proceedings of ASME PVP, 2003.

Ref. 7: Jo, J.C., Choi, Y.H. and Choi, S. K., November 2003, "Numerical Analysis of Unsteady Conjugate Heat Transfer and Thermal Stress for a PWR Pressurizer Surge Line Pipe Subjected to Thermal Stratification,"ASME Transaction J. of Pressure Vessel Technology. Vol. 125, pp. 467-474.

Ref. 8: O. Gélineau, M. Spérandio, J.-P. Simoneau, J.-M. Hamy, P. Roubin, 2002, "Validation of fast reactor thermomechanical and thermohydraulic codes : thermomechanical and thermal hydraulic analyses of a tee junction using experimental data", Final report of a co-ordinated research project, International Atomic Energy Agency, AIEA TECDOC-1318, Nov. 2002.

Ref. 9: O. Gélineau, C. Escaravage, J.-P. Simoneau, C. Faidy "High Cycle Thermal Fatigue: Experience and State of the Art in French LMFR, Proc. SMIRT16, 2001.

Ref. 10: J.-P. Simoneau H. Noé, B. Menant, "Large eddy simulation of sodium flow in a tee junction, comparison of temperature fluctuations with experiments", Proc. 8th Topical Mtg. Nuclear Reactor Thermal Hydraulics (NURETH-8), Kyoto, Japan, 1997.

3.17 Hydrogen Distribution

Relevance of the phenomenon as far as NRS is concerned

During the course of a severe accident in a water-cooled reactor, large quantities of hydrogen could accumulate in the containment.

What the issue is?

Detailed knowledge of containment thermal hydraulics is necessary to ensure the effectiveness of hydrogen mitigation methods. Condensation and evaporation on walls, pool surfaces and condensers needs to be adequately modelled, because the related mass and heat transfer strongly influence the pressure and mixture composition in the containment. For the Siemens containment design, the transient pressure rise causes certain explosion hatches to open (which defines the scenario). In addition, there is pressure loading to the structures. The mixture composition is very important, because it strongly determines the burning mode of hydrogen and the operation of the PARs (Passive Autocatalytic Recombiners).

What the difficulty is and why CFD is needed?

Containments have very large volumes and multi-compartments. The situation occurring in the context of a severe accident is also physically complex. A too coarse nodalisation will not only lose resolution, but will smear the temperature and velocity gradients through numerical diffusion. Temporal discretisation is also an important issue, as accident transients must be simulated over several hours, or even days, of physical time. From a physical point of view, the flow model must also take into account condensation (in the bulk or at the wall), together with heat transfer to the structures. Condensation models are not standard in CFD codes.

An additional, and significant, difficulty in the application of CFD to hydrogen distribution problems relates to the way in which reactor systems, such as recombiners, spray systems, sumps, etc., are taken into account. CFD simulations without such system/component models will not be representative of realistic accident scenarios in nuclear reactor containments.

What has been attempted and achieved/what needs to be done (recommendations)?

A State-of-the-Art report on this issue was proposed to the CSNI in 1995, and a group of experts convened to produce the document, which appeared finally in 1999. The twin objectives of the SOAR were to assess current capabilities to predict hydrogen distributions in containments under severe accident conditions, and to draw conclusions on the relative merits of the various predictive methods (lumped-parameter approaches, field codes, CFD). The report concentrates on the traditional containment codes (e.g. CONTAIN and GOTHIC), but acknowledges the future role of CFD-type approaches (e.g. GASFLOW, TONUS and ANSYS-CFX) to reduce numerical diffusion.

It was concluded that current lumped-parameter models are able to make relevant predictions of the pressure history of the containment and its average steam content, and that predictions of hydrogen distributions are adequate provided safety margins are kept high enough to preclude significant accumulations of sensitive mixtures, but that gas distribution predictions needed to serve as a basis for combustion analyses required higher resolution. The limits of the lumped-parameter approach have been demonstrated in a number of ISP exercises (notably ISP-23, ISP-29, ISP-35, and ISP-37). CFD-type approaches may be the better option for the future, but considerable validation and accumulation of experience were considered necessary before such tools could be reliably used for plant analyses. An on-going benchmark exercise, ISP-47, aims precisely at validating CFD codes for containment thermal-hydraulics, including hydrogen risk.

Hydrogen distribution occurring during a hypothetical station blackout (SBO) accident in the Korean next generation reactor APR1400 containment has been analysed using the 3-D CFD code GASFLOW (Ref. 6). Because the hydrogen was released into the in-containment refuelling water storage tank (IRWST) of the containment during the accident, the main concern was the hydrogen concentration and the possibility flame acceleration in the IRWST. In this study, design modifications were proposed and evaluated with GASFLOW in view of the hydrogen mitigation strategy.

Ref. 1: SOAR on Containment Thermalhydraulics and Hydrogen Distribution, NEA/CSNI/R(1999)16.

Ref. 2: A. Beccantini et al., “H2 release and combustion in large-scale geometries: models and methods”, Proc. Supercomputing for Nuclear Applications, SNA 2003, Paris, France, 22-24 September 2003.

Ref. 3: L. Blumenfeld et al., “CFD simulation of mixed convection and condensation in a reactor containment: the MICOCO benchmark”, Proc. 10th Int. Topical Meeting on Nuclear Thermal-Hydraulics, NURETH-10, Seoul, Korea, 5-9 October 2003.

Ref. 4: N.B. Siccama, M. Houkema, E.M.J. Komen “CFD analyses of steam and hydrogen distribution in a nuclear power plant”, IAEA-TECDOC-1379, 2003.

Ref. 5: International Standard Problem ISP-47 on Containment Thermal Hydraulics, Final Report, NEA/CSNI/R(2007)10.

Ref. 6: Jongtae Kim, Seong-Wan Hong, Sang-Baik Kim, Hee-Dong Kim, “Hydrogen Mitigation Strategy of the APR1400 NPP for a Hypothetical Station Blackout Accident”, Nuclear Technology, 150, 263-282 (2005).

Ref. 7: Jongtae Kim, Unjang, Lee, Seong-Wan Hong, Sang-Baik Kim, Hee-Dong Kim, “Spray effect on the behavior of hydrogen during severe accidents by a loss-of-coolant in the APR1400 containment”, *International Communications in Heat and Mass Transfer*, **33**, 1207–1216 (2006).

3.18 Chemical Reactions/Combustion/Detonation

Relevance of the phenomenon as far as NRS is concerned

Detonation and combustion in containments may lead to pressure rises which exceed the design specifications. There is also risk of localised overheating of structures in the case of standing flames.

What the issue is?

Although BWR containments are normally nitrogen inerted, which prevents hydrogen combustion and detonation, special attention has been addressed in recent years to possible leakage of hydrogen from the small overpressurised BWR containment to the reactor building, resulting in possible combustion and detonation, and providing a challenge for the containment integrity from outside.

For PWR containments that are not inerted, but which have some mitigation systems (recombiners, for example), local hydrogen concentrations can exceed the flammability limits, at least during some stages of the accident scenarios. Deflagrations, accelerated flames or even detonations are to be envisaged for some accident scenarios.

What the difficulty is and why CFD is needed to solve it?

Deflagrations are very complex phenomena, involving chemistry and turbulence. No adequate models exist to accurately describe deflagrations at large-scale and in complex geometries – but still, CFD combined with flame-speed-based deflagration models can provide significant insight into the dynamic loadings on the structures.

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. The interaction with the flow is limited to shock wave propagation – no turbulence models are necessary; in fact, it is generally sufficient to use the inviscid Euler equations. However, a fully compressible method must be used, typically a Riemann-type solver. Shock-wave simulations should account also for multiple reflections and superposition of the shock waves.

What has been attempted/achieved so far and what needs to be done?

A project has been carried out under NKS/SOS-2.3 for the calculation of containment loads (BWR) in the above postulated scenario. The CFD code FLUENT was used to calculate hydrogen distribution in the reactor building, DET3D (Karlsruhe) for the 3D detonation simulation, and ABAQUS for the structural analysis and evaluation of the loads. The conclusion of this study was that a more detailed analysis would be required to take into account the pressure decrease after the detonation.

There have been many applications of compressible CFD solvers to model detonations in large-scale geometries (e.g. the RUT experiments from the Kurchatov Institute), and also some calculations of fast deflagrations in a simplified reactor containment (EPR) were performed in the framework of the 5th FP Project HYCOM. H2 deflagration models and CFD codes were also evaluated in the 4th FP project HDC (Hydrogen Distribution and Combustion).

Ref. 1: NKS-61 Advances in Operational Safety and Severe Accident Research, VTT Automation, Finland, 2002.

Ref. 2: A. Beccantini, H. Paillère, "Modeling of hydrogen detonation for application to reactor safety", Proc. ICONE-6, San Diego, USA, 1998.

Ref. 3: U. Bielert et al., “Multi-dimensional simulation of hydrogen distribution and turbulent combustion in severe accidents”, Nuclear Engineering and Design, 209, 165-172 (2001).

Ref. 4: W. Scholtyssek et al., “Integral Large Scale Experiments on Hydrogen Combustion for Severe Accident Code Validation”, Final Report of HYCOM Project, Project FIKS-CT-1999-00004, to appear 2004.

Ref. 5: P. Pailhories, A. Beccantini, “Use of a Finite Volume scheme for the simulation of hydrogen explosions”, Technical meeting on use of CFD for safety analysis of reactor systems, including containment, Pisa, Italy, November 11-15, 2002.

3.19 Aerosol Deposition/Atmospheric Transport (Source Term)

Aerosol Deposition

Relevance of the phenomenon as far as NRS is concerned

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, aerosols would be released into the environment and pose a health hazard.

What the issue is?

The most conservative assumption is that all the fission-product aerosols 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 the aerosol dynamics and chemical behaviour in the containment.

What the difficulty is and why CFD is needed?

The global thermal-hydraulic response is primarily determined by the balance of flow of steam from the circuit and condensation. The overall behaviour is therefore governed by the thermodynamic state, and is well reproduced using simple lumped-parameter models with coarse nodalisation (one or two volumes), provided the boundary conditions are correctly imposed. Nonetheless, it should be realised that the adequacy of simple representations perhaps depends on simple geometry and well-defined conditions. Care should be taken when extrapolating such conclusions to the much more complex situations encountered in a real plant.

Consequently, the controlling phenomena for aerosol removal need to be assessed using a more rigorous treatment of the forces acting on the particles. To simulate particle motion, it is necessary to know the 3-D velocity field, and CFD is needed for this purpose. The goal is to determine the accuracy with which CFD tools are able to predict the lifetimes of aerosols circulating in a large volume, such as a real reactor containment. By tracking a number of such particles, statistical information on the actual deposition can be obtained, and from that a realistic estimate of release in the event of a containment breach.

What has been attempted and achieved/what needs to be done (recommendations)?

The PHEBEN-2 EU 5th Framework Programme aimed at improving the current analytical capability of realistically estimating power plant safety in the event of a hypothetical accident, based on the experimental information coming from PHEBUS-FP project. The PHEBUS-FP facility is operated at CEA Cadarache, and aims to investigate the key phenomena occurring in an LWR severe accident. The facility provides prototypic reactor conditions from which integral data on core degradation, fission product release, aerosol-borne transport and retention in the coolant circuit, and the aerosol dynamics and chemical behaviour in the containment may be obtained. A series of five experiments was carried out during the period 1993-2004, which simulated release and fission product behaviour for various plant states and accident situations. The definitive final document is currently in review, and expected to be released in 2008.

The experimental measurements from the PHEBUS tests, which must be remembered are of integral form, confirm the appropriateness of lumped-parameter, coarse-node models for calculating the global response of the containment, at least for the simple geometry and conditions considered in the tests. There is no indication that detailed models or CFD methods are needed to calculate the global behaviour, though such methods are being applied to scope the potential. In any event, such approaches would be necessary to calculate the hydrogen distribution, and may be needed for aerosol deposition in more realistic geometries. There is a definite lack of useful validation data of the type needed to validate the CFD models in open geometries.

Ref. 1: P. von der Hardt, A.V. Jones, C. Lecomte, A. Tattegrain, “The PHEBUS FP Severe Accident Experimental Programme”, Nuclear Safety, 35(2), 187-205 (1994).

Ref. 2: A. V. Jones et al., “Validation of severe accident codes against PHEBUS-FP for plant applications (PHEBEN-2)”. FISA-2001 EU Research in Reactor Safety, Luxembourg, 12-14 November 2001.

Ref. 3: A. Dehbi, “Tracking of aerosol particles in large volumes with the help of CFD”, Proceedings of 12th International Conference on Nuclear Engineering (ICONE 12), Paper ICONE12-49552, Arlington, VA, April 25-29, 2004.

Ref. 4: “State of the Art Report on Nuclear Aerosols”, NEA/CSNI/R(2009)5.

3.20 Atmospheric Transport (Source Term)

Relevance of the phenomenon as far as NRS is concerned

During a severe reactor accident, radioactive release to the atmosphere could occur, which may represent a health hazard for the installation workers and the surrounding population.

What the issue is?

Atmospheric release of nuclear materials (aerosols and gases) implies air contamination: on-site at first, and off-site with time. The atmospheric dispersion of such material in complex situations, such as the case of buildings in close proximity, is a difficult problem, but important for the safety of the people living and working in such areas. Dispersion models need meteorological fields as input; typical examples of such fields are velocity fields and characterisation of atmospheric thermal stability.

What the difficulty is and why CFD is needed to solve it?

CFD provides a method to build and run models that can simulate atmospheric dispersion in geometrically complex situations; however, the accuracy of the results needs to be assessed. Emergency situations, which lead to atmospheric release generally, involve two basic scales: on-site scale, where the influence of nearby buildings and source modelling are important phenomenon, and off-site scale (from a few kilometres to tens of kilometres), where specific atmospheric motions are predominant.

On-site atmospheric flows and dispersion are highly 3D, turbulent and unsteady, and CFD is a traditional approach to investigate such situations. Numerical modelling of building effects on the wind and dispersion pose several challenges. Firstly, computation of the flows around buildings requires knowledge of the characteristics of atmospheric boundary layers. In addition, knowledge of the mean wind speed and degree of atmospheric turbulence are also needed to accurately represent atmospheric winds, and the effects of the site, on dispersion. Secondly, topography of the configuration to be modelled is usually complex, especially in a Nuclear Power Plant, where closely spaced groups of buildings are commonplace, with different individual topologies, heights and orientations. Consequently, great challenges are encountered when discretising the computational domain. Thirdly, the flows are highly complex, having all the elements that modern fluid mechanics has not yet successfully resolved. The major challenge lies in turbulence modelling. The difficulty is associated with the fact that the flows are highly three-dimensional, being accompanied, almost without exception, by strong streamline curvature, separation, and vortices of various origin and unsteadiness.

What has been attempted/achieved so far and what needs to be done?

While most of the CFD applications to date have been focussed on the generation of wind fields, as input to dispersion models for the purposes of assessment or emergency preparedness, the utilisation of prognostic models in weather-related emergencies is beginning to be explored. Prognostic model forecasting on regional scales will play an important role in advising local agencies regarding emergency planning in cases of severe accidents. In addition, model output information, such as precipitation, moisture and temperature, are often necessary for predicting the movement of pollutants under complex meteorological conditions. For example, wet scavenging during precipitation is an important sink of airborne pollutants leading to the deposition of contaminants.

Workstation-based meso-scale models have recently been used to provide real-time forecasts at regional scales, for emergency response to locally-induced severe accidents. In regional response forecasting, meteorological forecasts of 3-48h are generated continuously, with nested grid resolutions of 1-20 km, centred at the specific site of interest. These locally-generated forecasts are available for dispersion calculations.

Ref. 1: Fast J.D., O'Steen B.L., Addis R.P. "Advanced atmospheric modelling for emergency response", J. Applied Meteor., 94, 626-649 (1995).

Ref. 2: Byrne C.E.I., Holdo A.E. "Effects of increased geometric complexity on the comparison between computational and experimental simulations", J. of Wind Eng. and Indus. Aerodyn., 73, 159-179 (1997).

Ref. 3: Ding F., Arya S.P., Lin Y.L. "Large eddy simulations of the atmospheric boundary layer using a new subgrid-scale model", *Environmental Fluid Mechanics*, **1**, 29-47 (2001).

3.21 Direct-Contact Condensation

This is a two-phase phenomenon, which is covered fully in the WG3 document.

Orientation

Some reactor designs feature steam discharge to cold-water pools. It is important to avoid steam by-pass in which vented steam may enter the vapour space above the pool and over-pressurise the confinement. The efficiency of the condensation process, and thermal mixing in the pool, may require detailed 3-D modelling using CFD.

3.22 Bubble Dynamics in Suppression Pools

This is a two-phase phenomenon, which is covered fully in the WG3 document.

Orientation

Again, and related to direct contact condensation, it is important to avoid steam by-pass into the vapour space to avoid over-pressurisation. For some advanced passive cooling system designs, containment gases are vented to suppression pools. Even with complete steam condensation, bubbles containing non-condensable gases remain, and to assess their ability to mix the water in the pool, and avoid stratification, requires detailed CFD modelling.

3.23 Behaviour of Gas/Liquid Interfaces

This is a two-phase phenomenon, which is covered fully in the WG3 document.

Orientation

In the two-fluid approach to two-phase flow modelling, as commonly employed in 1-D system codes and 3-D CFD codes, the two phases are treated as interpenetrating media. There are many instances of relevance to NRS in which the phases are physically separated and the phase boundary between them requires detailed resolution. Some examples are pressurised thermal shock (leading to thermal striping and cyclic-fatigue in structures), level detection in pressurisers, accumulators and the cores of BWRs (used for triggering ECC devices), and level swell in suppression pools. Given the 3-D nature of the flow regime, CFD methods, with direct interface-tracking capability, may be needed to accurately describe events. Some references regarding modelling approaches are given here.

Ref. 1: C. W. Hirt, B. D. Nichols, "Volume of Fluid method (VOF) for the dynamics of free boundaries", J. Comput. Phys., 39, 201-225 (1981).

Ref. 2: M. Meier, G. Yadigaroglu, B. L. Smith, "A novel technique for including surface tension in PLIC-VOF methods, Eur. J. Mech. B/Fluids, 21, 61-73 (2002).

Ref. 3: S. Osher, J. A. Sethian, "Fronts propagating with curvature-dependent speed: algorithms based on Hamilton-Jacobi formulations", J. Comput. Phys., 79, 12 (1988).

Ref. 4: J. A. Sethian, Level Set Methods, Cambridge University Press, Cambridge, UK, 1998.

3.24 Special Considerations for Advanced Reactors

Coolability of radial reflector of APWR

Relevance of the phenomenon as far as NRS is concerned

Insufficient cooling of the radial reflector causes thermal deformation of the reflector blocks, which results in formation of a gap between blocks. A leak flow through the gap decreases the core flow rate, and may raise the temperature of the reactor core.

What the issue is?

The radial reflector consists of a stack of eight SUS304 blocks, in which many holes are installed to cool the reflector blocks, which become hot due to the heat generation of gamma rays. A large amount of the coolant which enters in the reactor vessel from the inlet nozzles flows up into the core region, and a small part of that flows into the radial reflector (Figs. 1,2) If the coolant flow rate into the radial reflector falls short, or becomes uneven circumferentially, the temperature of the coolant rises and the coolant may possibly boil (Fig.3).

Since the reflector block is not symmetrical and the heat generation of gamma rays is not spatially uniform, the temperature distribution of the reflector block becomes uneven, and a deformation of the block due to the differences of the thermal expansion, produces a gap between the adjacent blocks. Consequently, the gaps cause bypass flow from the reactor core side into the neutron reflector.

What the difficulty is and why CFD is needed to solve it?

Evaluation of the temperature distribution in the reflector blocks with sufficient accuracy needs a detailed description of the coolant flow rate into the reflector. The details of this flow depend on the coolant flow field in the reactor vessel, and the flow field in lower plenum is complicated because of the asymmetrical arrangement of the structures. CFD is therefore the only effective tool for evaluating the coolant flow field in the reactor vessel.

What has been attempted/achieved so far and what needs to be done?

The three-dimensional flow field in the reactor vessel, and the distribution of the coolant flow rate into the radial reflector, have been evaluated using the CFD code uFLOW/INS with the standard k-epsilon turbulent model. The uFLOW/INS code has been validated against experimental data from a 1/5-scale APWR experiment. Evaluation of the coolability of the radial reflector needs the correct calculation of the flow rates through the very small cooling holes installed in the reflector blocks. A technique is required for modelling these small holes without substantially increasing the total number of grid points used for the calculational domain.

Ref. 1: T. Morii "Hydraulic flow tests of APWR reactor internals for safety analysis", Benchmarking of CFD Codes for Application to Nuclear Reactor Safety, Garching, Munich, Germany 5-7 September 2006.

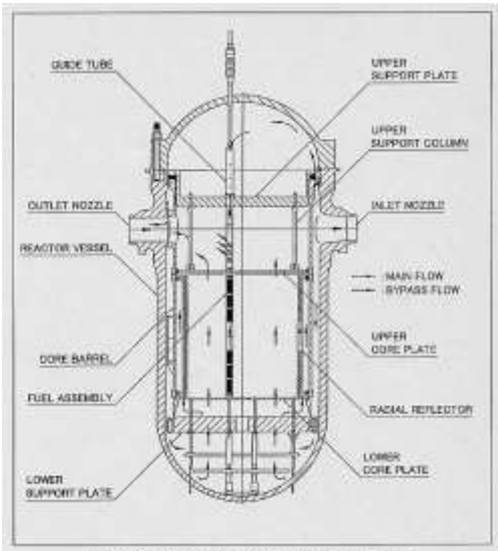


Figure 1 APWR Reactor Vessel and Internals

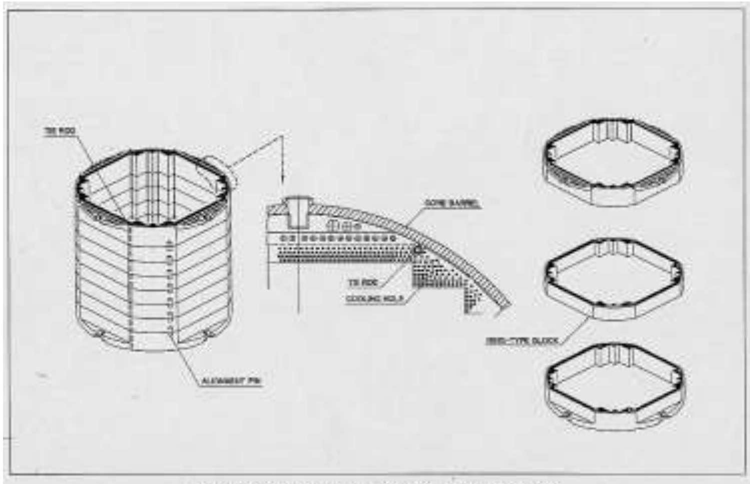


Figure 2 Configuration of APWR Radial Reflector

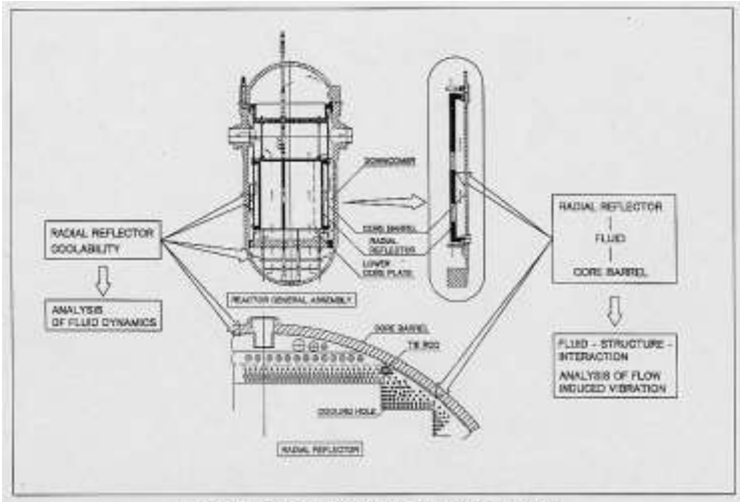


Figure 3 Flow Behavior and Associated Analyses

3.25 Flow induced vibration of APWR radial reflector

Relevance of the phenomenon as far as NRS is concerned

Flow-induced vibrations of the radial reflectors in APWRs could result in fretting, and possibly rupture, of the fuel pin cladding

What the issue is?

If the core barrel is vibrated by the turbulent flow in the downcomer, it vibrates the radial reflector through the water between them (Fig.4). If the radial reflector vibrates, the grid of the outermost fuel bundles may make contact with it, and when the grid vibrates, the fuel clad may be worn out.

What the difficulty is and why CFD is needed to solve it?

In order to evaluate the vibration of the radial reflector with sufficient accuracy, it is necessary to calculate the pressure fluctuations of the turbulent flow in the downcomer correctly, which is the driving force of the vibration. The following two methods are available for using CFD for evaluating the vibration between fluid and structure; the latter method is more practical.

(1) The vibration between fluid and a structure is calculated directly by the coupled use of a CFD code and a structural analysis code, using the moving boundary technique.

(2) The vibration between fluid and a structure is calculated by the structural analysis code, modelling the water between the core barrel and the radial reflector as simply an additional mass, and imposing the downcomer pressure fluctuations calculated by the CFD code as load conditions.

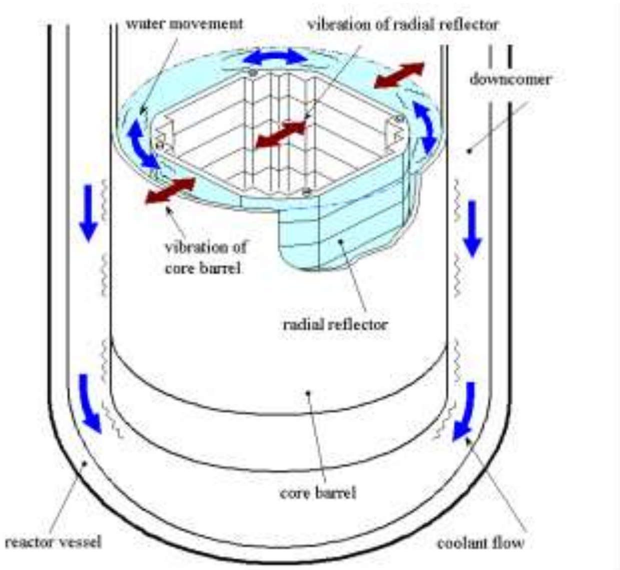


Figure 4 Flow-induced vibration of radial reflector

What has been attempted/achieved so far and what needs to be done?

The vibration between fluid and structures has been calculated using the structural analysis code FELIOUS. The distribution of the downcomer fluid pressure fluctuations, which is used as the load conditions in the input data of the FELIOUS code, is obtained from a statistical analysis of the experimental data of the 1/5-scale APWR test facility. Moreover, the 3-dimensional transient analysis of the turbulent flow in the downcomer has been carried out using a CFD code with LES (Large Eddy Simulation) turbulence model, and the calculated results have been compared with the above mentioned experimental data. The application of the LES model with high accuracy to the large calculation system of several orders of magnitude difference in scale is needed.

Ref. 1: F. Kasahara, S. Nakura, T. Morii, Y. Nakadai, “Improvement of hydraulic flow analysis code for APWR reactor internals”, CFD Meeting in Aix-en-Provence, May 15-16, 2002, NEA/CSNI/R(2002)16.

3.26 Natural circulation in LMFBRs

Relevance of the phenomenon as far as NRS is concerned

Current LMFBR designs often feature passive devices for decay-heat removal. It is necessary to demonstrate that the system operates correctly under postulated accident conditions.

What the issue is?

Decay heat removal using natural circulation is one of the important functions for the safety of current LMFBRs. For example, DRACS (Direct Reactor Auxiliary Cooling System) has been selected for current designs of the Japanese Demonstration Fast Breeder Reactor. DRACS has *Dumped Heat Exchangers* (DHXs) in the upper plenum of the reactor vessel. Cold sodium provided by the DHX covers the reactor core outlet, and also produces thermal stratification in the upper plenum (Fig.1). In particular, the decay heat removal capability has to be assured for the total blackout accident in order to achieve high reliability.

What the difficulty is and why CFD is needed to solve it?

The cold sodium in the upper plenum can penetrate into the gap region between the subassemblies due to negative buoyancy, and enhances the natural convection in these gap regions. Analyses of natural circulation tests in the Japanese experimental reactor JOYO revealed that heat transfer between subassemblies, i.e. inter-subassembly heat transfer, reduced subassembly outlet temperatures for the inner rows of the core. CFD is effective in evaluating the complex flow field caused by natural convection in the LMFBR reactor vessel.

What has been attempted/achieved so far and what needs to be done?

The three-dimensional flow field and temperature distribution of sodium in the reactor vessel have been evaluated by JNC (Japan Nuclear Cycle Development Institute) using the CFD code AQUA.

The three-dimensional natural convection in the reactor vessel, coupled with the one-dimensional natural circulation in the loops, have been evaluated simultaneously by JAPC (Japan Atomic Power Company) using a CFD code combined with a system code.

Ref. 1: H. Kamide, K. Hayashi, T. Isozaki, M. Nishimura, “Investigation of Core Thermohydraulics in Fast Reactors - Interwrapper Flow during Natural Circulation”, *Nuclear Technology,* **133**, 77-91 (2001).

Ref. 2: H. Kamide, K. Nagasawa, N. Kimura, H. Miyakoshi, “Evaluation Method for Core Thermohydraulics during Natural Circulation in Fast Reactors (Numerical Predictions of Inter-Wrapper Flow)”, JSME International Journal, Series B, Vol.45, No.3, 577-585, 2002.

Ref. 3: Watanabe et al., “Study on Natural Circulation Evaluation Method for a large FBR”, Proc. NURETH-8 Conference, Kyoto September 30 - October 4, 1997.

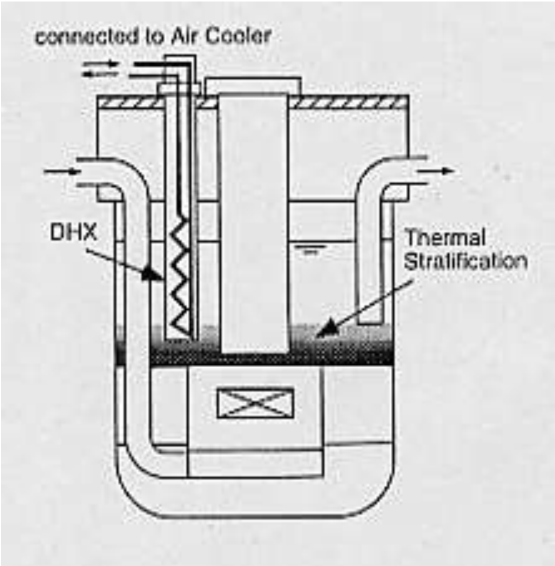


Fig. 1 Schematic of DRACS

3.27 Natural Circulation in PAHR (Post Accident Heat Removal)

Relevance of the phenomenon as far as NRS is concerned

Following a loss of core geometry as a consequence of a severe accident in an LMFBR, the availability of the decay heat removal systems have to be guaranteed to prevent possible melt-through of the reactor vessel.

What the issue is?

After a core disruptive accident in an LMFBR, molten core material is quenched and fragmented in the sodium and settles to form a debris bed on structures in the reactor vessel. If the decay heat generated within the debris bed is not removed over a long period of time, the debris bed could melt again, and cause failure of the reactor vessel.

What the difficulty is and why CFD is needed to solve it?

Decay heat in the debris bed is removed by natural convective flows passed through several leak paths which do not exist under normal operation conditions in current designs of Japanese Demonstration Fast Breeder Reactor (Fig.2). CFD methods are effective in evaluating the above-mentioned complicated natural circulation flow to high accuracy.

What has been attempted/achieved so far and what needs to be done?

The 3-dimensional natural circulation flow in the above-mentioned situation has been evaluated using a state-of-the-art CFD code. (There is no open report).

Ref. 1: K. Satoh et al., “A study of core disruptive accident sequence of unprotected events in a 600MWe MOX homogeneous core”, Proc. of Int. Conf. on Design and Safety of Advanced Nuclear Power Plants, Tokyo, Japan, 25-29 October 1992.

Ref. 2: K. Koyama et al., “A study of CDA sequences of an unprotected loss-of flow event for a 600MWe FBR with a homogeneous MOX core”, IWGFR/89 IAEA Technical Committee Meeting on Material-Coolant Interactions and Material Movement and Relocation in Liquid Metal Fast Reactors, O-arai, Ibaraki, Japan, 6-9 June 1994.

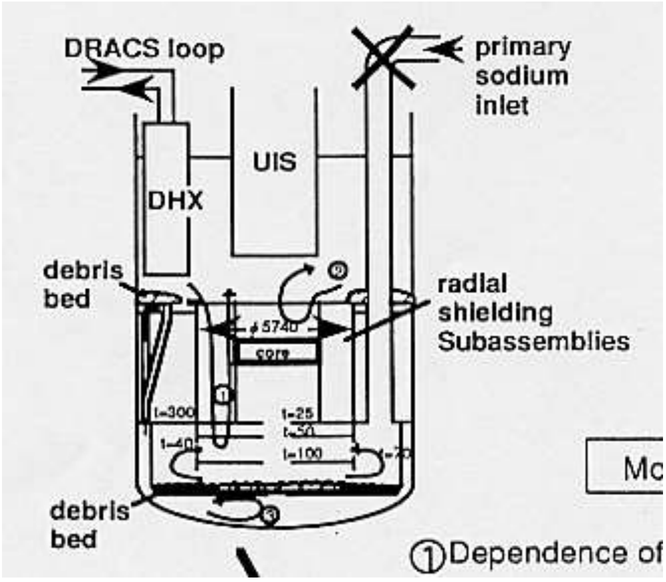


Figure 2: Schematic of the Demonstration Fast Reactor

3.28 Gas Flow in the Containment following a Sodium Leak

Relevance of the phenomenon as far as NRS is concerned

The sodium coolant used in LMFBRs is a hazardous material, and adequate precautions have to be made if a spill occurs.

What the issue is?

Liquid sodium has preferable characteristics as a coolant in LMFBRs from both the neutronics and thermal-hydraulics viewpoints. On the other hand, liquid sodium will chemically react with oxygen or water if it leaks out of heat transport system. For the safety of the LMFBR plants, it is important to evaluate the consequence of possible sodium combustion.

What the difficulty is and why CFD is needed to solve it?

Leaked sodium may break up into small droplets of various diameters. In an air atmosphere, the droplets burn as they fall. This is designated as spray combustion. The unburned sodium collects on the floor of the reactor building, and pool combustion may ensue (Fig.3).

In order to evaluate the spray combustion rate with sufficient accuracy, it is necessary to evaluate the amount of oxygen which flows around the sodium droplets. The amount of oxygen depends on the gas flow in the room caused by the motion of sodium droplets, and the temperature/concentration stratification.

On the other hand, in order to estimate the pool combustion rate with sufficient accuracy, it is necessary to evaluate the amount of oxygen which flows to the sodium pool surface. This depends on the natural convection flow generated on the hot pool surface. A CFD code is effective in evaluating this gas flow.

What has been attempted/achieved so far and what needs to be done?

The CFD code AQUA-SF has been developed by JNC (Japan Nuclear Cycle Development Institute) to evaluate spatial distributions of gas temperature and chemical species. The code includes the spray combustion model and a flame-sheet pool combustion model.

Ref. 1: A. Yamaguchi, T. Takata, Y. Okano, "Numerical Methodology to Evaluate Fast Reactor Sodium Combustion", Nuclear Technology, 136, 315-330, (2001).

Ref. 2: T. Takata, A. Yamaguchi, I. Maekawa, "Numerical Investigation of Multi-dimensional characteristics in sodium combustion", *Nuclear Engineering and Design*, **220**, 37-50 (2003).

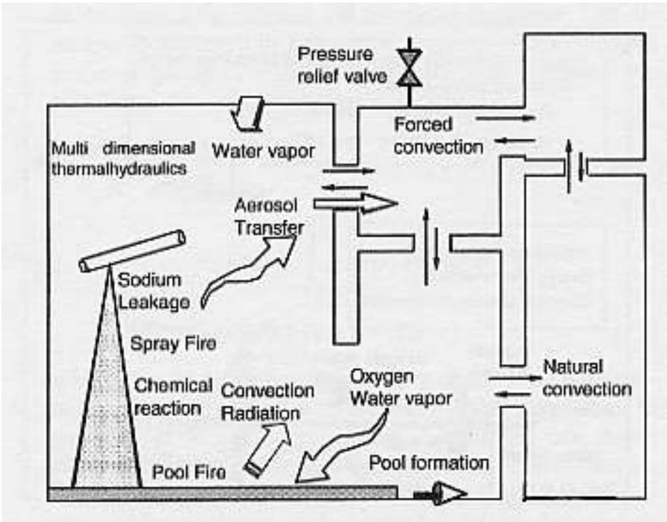


Figure 3: Computational Models for the SPHINCS Program

3.29 AP600, AP1000 and APR1400

Relevance of the phenomenon as far as NRS is concerned

The AP600 is a 2 loop PWR, designed by Westinghouse, with passive safeguard systems. The passive safety systems, such as core make-up tanks and the passive, residual-heat-removal heat exchanger, depend on gravity. The availability and functionality of these components has been confirmed as part of the licensing procedures. However, certain aspects of the operation involve 3-D flow behaviour, and there is scope for CFD to be employed to improve efficiency and reduce the degree of conservatism in the design.

What the issue is?

The AP600 has several passive system components, and thermal-hydraulic phenomena relating to these components will occur during accidents or transients: thermal stratification in the core makeup tank (CMT), downcomer and cold legs, condensation and convection in the in-containment refuelling water storage tank (IRWST), and so on.

In the IRWST, three-dimensional thermal convection due to the heat transfer from the passive residual heat removal (PRHR) heat exchanger, and the condensation of steam from the automatic depressurisation system (ADS), are both important for cooling of the primary system.

Thermal stratification in cold legs is one of the significant phenomena under some small-break LOCA conditions after the termination of the natural circulation through the steam generators. In the loop where the PRHR system is connected, the fluid in the cold leg is a mixture of the draining flow from the steam generator U-tubes and the discharge from the PRHR heat exchanger in low-temperature IRWST, and becomes significantly colder than the downcomer liquid. The relatively warmer downcomer liquid intrudes along the top of the cold leg. In contrast, in the loop with the CMT, the cold-leg liquid is kept at a higher temperature than the downcomer liquid temperature, since the CMT water is injected into the downcomer through the direct vessel injection (DVI) line, and the downcomer liquid intrudes along the bottom of the cold leg. In both cases, a counter-current flow is established as well as the thermal stratification. In case of cold-leg break LOCAs, the thermal stratification in the cold legs has an effect upon the discharge flow rate from the break point, and thus the system response.

What the difficulty is and why CFD is needed to solve it?

Three-dimensional convection in a tank and counter-current thermal stratification in legs are difficult phenomena to model using system analysis codes based on one-dimensional components. The difference of discharge from a break point due to the difference of orientation is not generally accounted for. The system behaviour, however, is associated with these local phenomena, and a CFD approach is necessary for safety evaluation of new types of components and reactors.

What has been attempted/achieved so far and what needs to be done?

Three-dimensional calculations for single-phase flows are possible using commercial CFD codes. The cold-leg flow, however, becomes a two-phase mixture under some conditions, and is much influenced by the system response. The flow in the IRWST is also related strongly to the system response. Detailed three-dimensional calculations of single- and two-phase flows are necessary at the same time with, or in the framework of, the system analyses.

Ref. 1: http://www.iaea.or.at/programmes/ne/nenp/nptds/newweb2001/simulators/cti_pwr/pwr_ap600_overview.pdf

Ref. 2: I.S. Kim and D.S. Kim, “APR1400: Evolutionary Korean Next Generation Reactor”, Proc. ICONE-10, Arlington, USA, April 14-18, 2002.

Ref. 3: C.-H. Song, W.P. Baek, J.K. Park, “Thermal-Hydraulic Tests and Analyses for the APR1400’s Development and Licensing”, J. Nuclear Eng. & Technology 39(4), Aug. 2007.

3.30 SBWR, ESBWR and SWR-1000

Relevance of the phenomenon as far as NRS is concerned

Evolutionary-design reactor systems often feature passive decay-heat removal systems, including passive decay heat removal from the containment in the event of a LOCA. The coupling of the primary circuit and containment response is a new concept, and needs to be thoroughly understood in order to ensure safe operation of the reactor under such conditions.

What the issue is?

The phenomena to be investigated involve mixing and transport of the containment gases — steam and incondensables (nitrogen and, in the case of severe accidents involving core degradation, possibly also hydrogen) — and condensation of the steam on cold surfaces and/or water pools.

What the difficulty is and why CFD is needed?

Generally, in all the above cases, decay heat removal involves complex mixing and transport of two-component/two-phase flows in complex geometries. The numerical simulation of such behaviour requires the use of sophisticated modelling tools (i.e.? CFD) because of the geometric complexities and the inherent 3-D behaviour, together with the development of reliable and appropriate physical models.

The principles, which reflect the need for advanced tools, may be illustrated with reference to the schematic of the ESBWR shown in Fig. 4a. The Drywell is directly connected to Passive Containment Cooler (PCC) units, which sit on the containment roof. The steam condensed in these units is fed back to the Reactor Pressure Vessel (RPV), while any uncondensed steam, together with the nitrogen which originally filled the Drywell atmosphere, is vented to the Suppression Pool. Clearly, partial condensation of steam and stratified conditions in the pool are both unfavourable, leading to excess pressure in the chamber. It is therefore important to understand the condensation and mixing phenomena which occur in the pool. To accurately represent the dynamics of the bubble expansion and break-up, CFD, in combination with an interface tracking procedure (e.g.? VOF or LS) is required.

Following break-up of the primary discharge bubble into smaller bubbles, it is no longer convenient to explicitly describe the liquid/gas interface, because of its disjointedness and complexity. Consequently, an Euler/Euler, two-fluid approach has been followed, with the water acting as the continuous medium and the bubbles representing the dispersed phase. A full description of the bubble dynamics, and the stirring of the water in the pool to break up stratified layers, will encompass CFD with two-phase flow and turbulence models.

In the SWR-1000 (Fig. 4b), containment condensers are employed. One condenser under consideration is a cross-flow, finned-tube heat exchanger with steam condensation outside the tubes and water evaporation within. The tubes are slightly inclined and staggered (Fig. 5). The performance of such finned tube containment condensers can be investigated at small and medium scale, but the scaling factors remain uncertain for a full-sized unit. CFD offers an opportunity to analyse the full-scale situation cheaply and efficiently, using data from smaller tests to validate the models.

What has been attempted and achieved/what needs to be done (recommendations)?

Aspects of the issues alluded to above have been tackled using CFD methods in the context of the EU shared-cost actions TEMPEST, IPSS, INCON and ECORA. In addition, CFD has been used to model the mock-up experiments carried out in the PANDA facility. Considerable modelling effort has been expended on condensation in the presence of incondensables, interface tracking of gas-discharge bubbles and bubble plumes in suppression pools. Requiring more attention is the extension of the two-phase CFD models for condensation and turbulence.

Ref. 1: S. Rao, A. Gonzalez, 1998, "ESBWR: Using Passive Features for Improved Performance and Economics", Proc. Nucl. Conf., Nice, France, 26-28 Oct. 1998.

Ref. 2: G. Yadigaroglu, 1999, "Passive Core and Containment Cooling Systems: Characteristics and State-of-the-Art", Keynote Lecture, NURETH-9, 3-8 Oct., 1999.

Ref. 3: N. S. Aksan, and D. Lubbesmeyer, “General Description of International Standard Problem 42 (ISP-42) on PANDA Tests”, Proc. Int. Conf. ICONE9, Nice, France, April 8-12, ASME/JSME/SFEN, 2001.

Ref. 4: Wickers, V. A., et al., 2003. Testing and Enhanced Modelling of Passive Evolutionary Systems Technology for Containment Cooling (TEMPEST), FISA 2003 Conference, EU Research in Reactor Safety, Luxembourg, 2003.

Ref 5: Andreani, M., Putz, F., Dury, T. V., Gjerloev, C. and Smith, B. L., 2003. On the application of field codes to the analysis of gas mixing in large volumes: case studies using CFX and GOTHIC. Annals of Nuclear Energy, 30, 685-714.

Ref. 6: Yadigaroglu, G, Andreani, M., Dreier, J. and Coddington, P., 2003. Trends and needs in experimentation and numerical simulation for LWR safety, *Nucl. Eng. Des.*, **221**, 205-223.

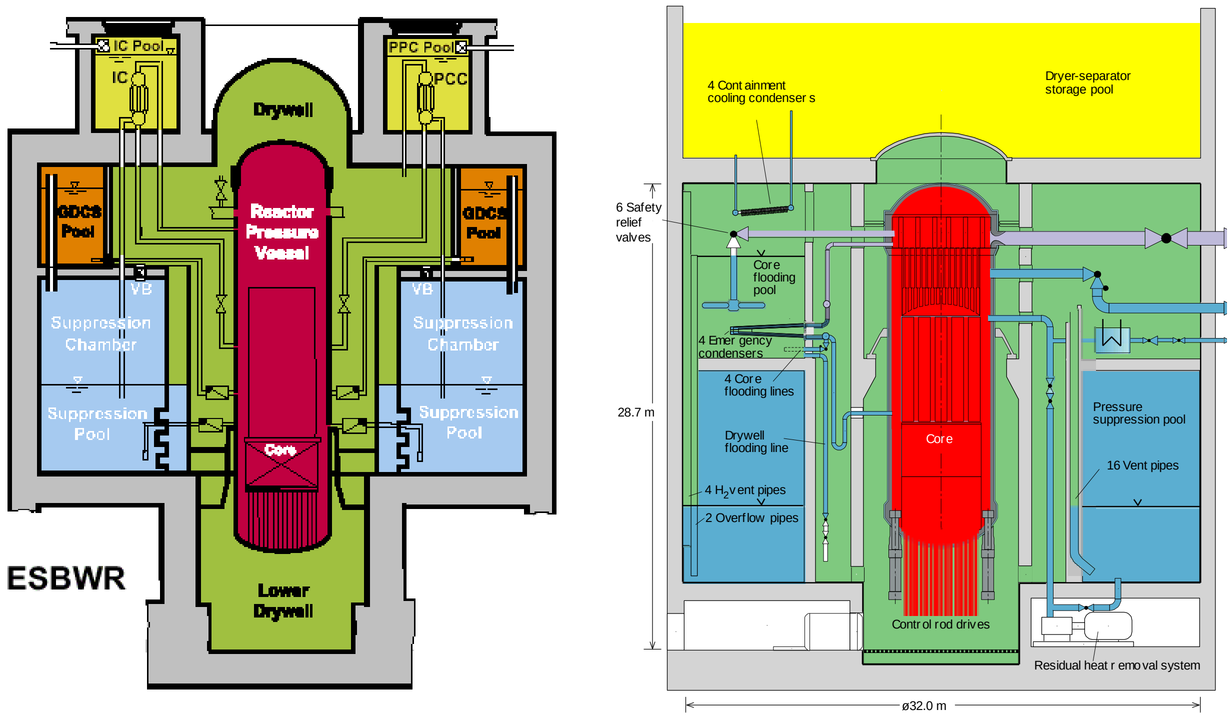


Figure 4: Two Evolutionary Reactor Designs: (a) ESBWR, (b) SWR-1000

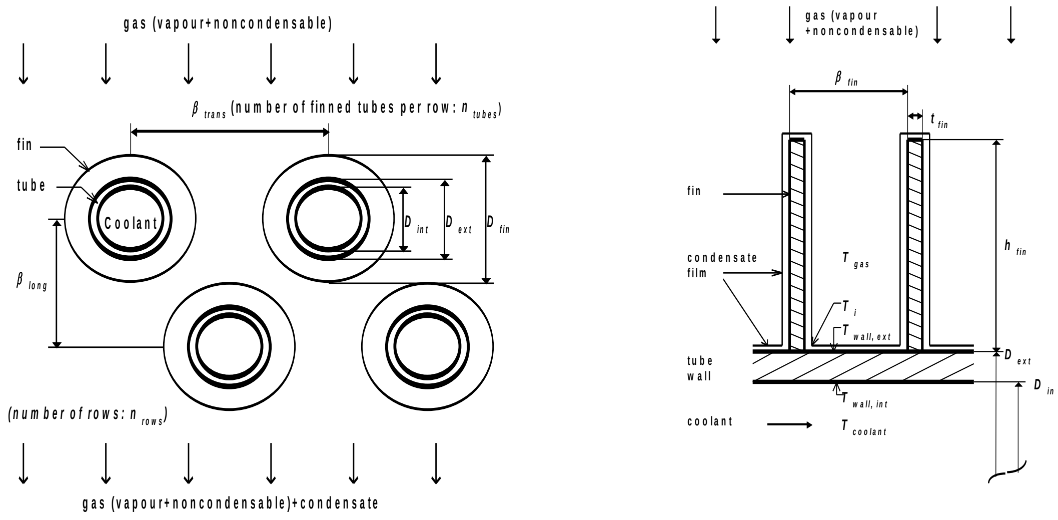


Figure 5: Bundle and Finned Tube geometries

3.31 High Temperature Gas-Cooled Reactor

Relevance of the phenomenon as far as NRS is concerned

The relevant part of the HTGR as far as NRS is concerned may be the containing vessel as well as the whole circuit, including the lower and upper plena, the power conversion system (for direct Brayton Cycle) and the core. One principal concern is that, for most of the accident scenarios for these reactors, safety relies on a passive system of residual power release. For other cases, such as “abrupt power rise” and even LOCA, NRS relies on the beneficial effect of thermal core inertia (graphite), the eventual power release being ensured by radiation transfer from the core to the vessel walls. This perspective relies on the behaviour of the core at high temperatures (Triso-particle).

What the issue is?

The issues depend on the precise part of the reactor under consideration.

1. *Primary Loop Ducts*. The NRS scenario may concern breaks in ducts that may lead to air ingress and possible air/graphite interaction.
2. *Containing Vessel*. The basic issue here is to precisely determine the global heat transfer between the core and the vessel walls, resulting from both natural convection and radiation. The two main issues are to check the capability of the system to remove all power while preserving the vessel integrity, and to identify the hot spots.
3. *Lower Plenum*. One of the basic issues is the reliance placed on the calculation of the flow behaviour in the lower plenum: for example, in column matrices (Ref. 1). The main physics relies on the capability of the system to mix flows of different temperatures to avoid temperature fluctuations on support structures, as well as at the turbine inlet.
4. *Upper Plenum.* First issue is related to Item 1 (heat release through radiation process), and the second issue concerns temperature fluctuations on internal structures.

5. *Turbine*. First issue is connected with Item 2 (temperature heterogeneity at the inlet for nominal and accident scenarios). Second issue concerns the temperature of the blades and disks. Indeed, these structures may not be cooled in some designs. For all the transients where these structures are not cooled, the question of thermal constraints arises. Other issues concern the dynamical behaviour: pressure variation, rotating speed variation, etc.
6. *Compressor*. Particular regimes such as stall or surge in the case of depressurisation may be of concern.
7. *Heat exchanger*. Firstly, the water exchangers are the only cold source of the primary loop. They should be checked for many transient situations: e.g. loss of load, pre-cooler failure, etc. NRS scenarios may also concern secondary loop water ingress. Secondly, the heat recuperator is submitted to temperature and pressure fluctuations at inlet.
8. *Core*. The core is subject to the usual problems, such as power rise, LOCA, etc.

What the difficulty is and why CFD is needed?

Geometries are complex, and it is difficult to make simplifications to ease modelling. Transients (which may be short or very long) involve multi-physics phenomena: CFD has to be employed in combination with conjugate heat transfer, radiation and neutronics coupling, for example, and the flow regimes are varied and complex (from incompressible to compressible, from laminar to turbulent – and sometimes with relaminarisation – and from forced to mixed and natural convection).

CFD is required, or is at least preferable, in the following circumstances.

- Where real three dimensional flows occur, which is typically the case for:
  - the core in accident situations (tube plugging or power rise);
  - the lower plenum, since asymmetrical flow develops due to the position of the outlet;
  - the heat exchanger, though here the case for CFD is questionable, since such a component can be taken into account only at the system level; however, a precise description of the phenomena may require CFD.
- Where complex flows develop in situations in which details of local quantities or local phenomena are needed. This is the case for:
  - the turbine, where local information about hot spots is required;
  - the compressor, where stall prediction is an issue;
  - generally, where local values are needed for the determination of hot spots.
- Even if the global behaviour in the upper plenum may be described as a component through a 0-D system approach, CFD may produce a more accurate description of the mixing processes occurring as a result of turbulence action.
- The precise description of local effects may be of relevance in the case of air ingress prediction, thermal fatigue (the GCR counterpart of the PWR tee-junction or thermal shock problem).

What has been attempted and achieved/what needs to be done (recommendations)?

Pioneering simulations concerning flows around lower plenum columns, and flows in some regions of the core, have been conducted at CEA (Ref. 1 to 5).

Ref. 1: Tauveron, N. “Thermal fluctuations in the lower plenum of a high temperature reactor”, Nuclear Engineering and Design, 222, 125-137 (2003).

Ref. 2: M. Elmo, O. Cioni, Low Mach number model for compressible flows and application to HTR, Nuclear Engineering and Design 222, 2003

Ref. 3: E. Studer et al., “Gas Cooled Reactor Thermal-Hydraulics using CAST3M and CRONOS2 codes”, Proc. 10th Int. Topical Meeting on Nuclear Thermal-Hydraulics, NURETH-10, Seoul, Korea, 5-11 October 2003.

Ref. 4: O. Cioni, M. Marchand, G. Geffraye, F. Ducros, “3D thermal hydraulic calculations of a modular block type HTR core”, Nuclear Engineering and Design 236, 2006

Ref. 5: O. Cioni, F. Perdu, F. Ducros, G. Geffraye, N .Tauveron, D. Tenchine, A. Ruby, M. Saez Multi-scale analysis of gas Cooled Reactors through CFD and system codes, ENC'2005, Versailles, France.

3.32 Sump Strainer Clogging

Relevance of the phenomenon as far as NRS is concerned

In a loss-of-coolant accident (LOCA) in a Pressurised Water Reactor (PWR), the two-phase jet flow from the break could strip off thermal insulation from the piping system and wash down the broken and fragmented debris to the sump screens. A total, or even partial, blockage of the screens could seriously inhibit the effectiveness of the decay-heat removal system.

What the issue is?

The particle load on the strainers results in an increased pressure drop, and hence decreased mass flow rate through the strainers. Sedimentation of the insulation debris on the screens, and its possible re-suspension and transport in the sump water flow, need to be accurately quantified to ensure continuous heat removal capability. This involves estimating the mass of fibre material deposited on the screens for a specified geometry of the reactor sump, and of the mass dragged on by the water flow. Ultimately, the mass transport of coolant determines the efficiency of the core cooling process.

What the difficulty is and why CFD is needed to solve it?

During the long-term core cooling operation following the LOCA, the water falls from the break from a height of several meters onto the sump water surface. During its transit, the water stream will mix with the air around. Air bubbles and released materials will be transported to the sump. The jet-induced flow into the sump will influence the transport of fibrous insulation material to the sump strainer, and consequently the head-loss across the strainer. CFD is able to calculate the main flow characteristics during the plunging jet situation. The establishment of a large swirling flow in the sump water caused by the entrained air can be reproduced using CFD, as can the transport of the fibrous material. The swirling flow patterns, which directly affect the fibre deposition properties, are three-dimensional phenomena, and cannot be captured using a traditional system-code approach.

What has been attempted/achieved so far and what needs to be done?

A joint research project has been set up between the University of Applied Science Zittau/Görlitz (HZGR) and the Helmholtz-Zentrum Dresden-Rossendorf (HZDR), involving an experimental investigation of particle transport phenomena (HZGR), and the development of appropriate CFD models for its simulation (HZDR). In the project, the fragmentation at prototypic thermal-hydraulic conditions, the transport behaviour of the fibres in a turbulent water flow, and the deposition and possible re-suspension of fibres have all been investigated. In addition, a numerical “strainer model” has been developed, the fibre behaviour being investigated for conditions of a plunging jet in a large pool. In a later part of the project, the scope was extended to include the effects of the presence of fibres in the core region, and consideration was also given to the chemical phenomena associated with them.

Ref. 1: Grahn, A.; Krepper, E.; Alt, S.; Kästner, W. “Modelling of differential pressure buildup during flow through beds of fibrous materials”, *Chemical Engineering & Technology*, **29**(8), 997-1000 (2006).

Ref. 2: Grahn, A.; Krepper, E.; Alt, S.; Kästner, W. “Implementation of a strainer model for calculating the pressure drop across beds of compressible, fibrous materials”, *Nuclear Engineering and Design*, **238**, 2546-2553 (2008).

Ref. 3: Grahn, A.; Krepper, E.; Weiß, F.-P.; Alt, S.; Kästner, W.; Kratzsch, A.; Hampel, R. “Implementation of a pressure drop model for the CFD simulation of clogged containment sump strainers”, *Journal of Engineering for Gas Turbines and Power - Transactions of the ASME*, **132**, 082902 (2010).

Ref. 4: Höhne, T.; Grahn, A.; Kliem, S.; Weiss, F.-P, “CFD simulation of fibre material transport in a PWR under loss of coolant conditions”, *Kerntechnik*, **76**, 39-45 (2011).

Ref. 5: Krepper, E.; Cartland-Glover, G.; Grahn, A.; Weiss, F.-P.; Alt, S.; Hampel, R.; Kästner, W.; Seeliger, A., “Numerical and experimental investigations for insulation particle transport phenomena in water flow”, *Annals of Nuclear Energy*, **35**, 1564-1579 (2008).

Ref. 6: Krepper, E.; Weiß, F.-P.; Alt, S.; Kratzsch, A.; Renger, S.; Kästner, W. “Influence of air entrainment on the liquid flow field caused by a plunging jet and consequences for fibre deposition”, *Nuclear Engineering and Design*, 241, 1047–1054 (2011).

4. DESCRIPTION OF EXISTING ASSESSMENT BASES

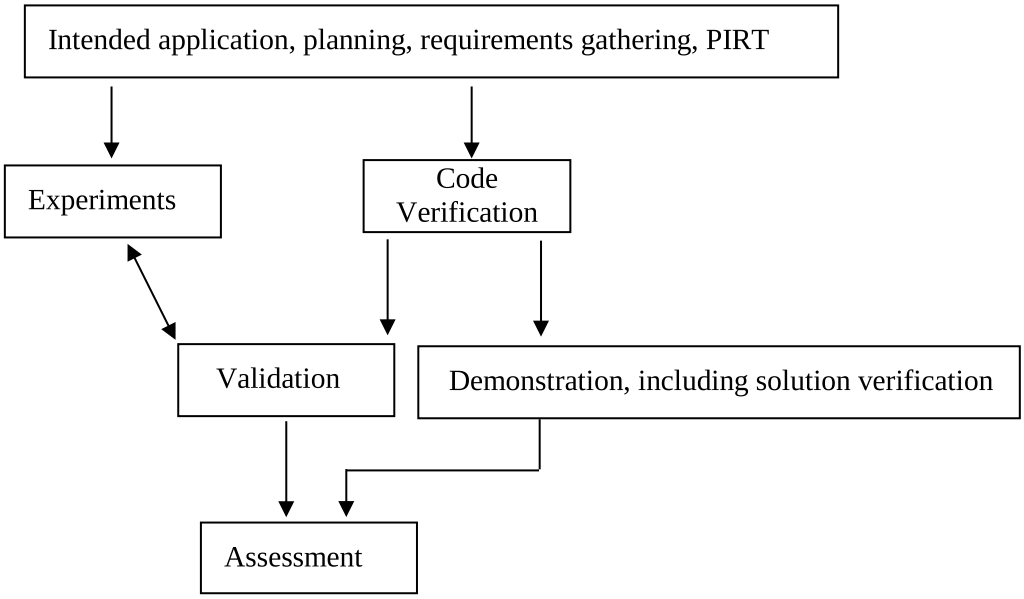
Major sources of information identified by the Group are elaborated below under appropriate section headings. In addition, in summary form, references to documents available from the NEA/CSNI and elsewhere are collected at the end of the section.

Some of the web sites referenced below allow free access to data for code validation, they sometimes propose CFD reference calculations, and they 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 at least demonstrate the seriousness of the commitment to "quality and trust" in CFD, and the concept could be expanded to serve the nuclear community also.

To be precise with the definition, *assessment* is defined here as an application-specific process based on three principal steps:

1. Verification (solving the equations correctly);
2. Validation (solving the correct equations); and
3. Demonstration (i.e., demonstrating the capability to solve a given class of problems).

This process is seen schematically in the Figure below.



An assessment matrix for a given application should therefore be composed of three groups of items (particular matrices):

4. “Exact” solutions and corresponding CFD calculations;
5. Validation experiments and corresponding CFD simulations; and
6. Demonstration CFD simulations, and possibly prototype experiments.

The following general statement can therefore be made:

“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 for the intended application of the code”.

As a consequence, a separate assessment matrix should be prepared for every selected nuclear safety issue where CFD simulation can be beneficial (see Chapter 3). This is a very demanding task. Fortunately though, many items (particular matrices) will be the same in the majority of such groups of matrices associated with different applications, since the same numerical algorithm and physical models will often be used.

Whereas verification should be performed mainly by code developers, validation and demonstration are strictly application-dependent and must therefore be performed, or at least overseen, by users. Validation and demonstration are the principal themes of this document. A review of several available general-purpose databases comprising experimental data is presented below under appropriate sub-headings. Then, specific application areas, namely boron dilution, pressurized thermal shocks, thermal fatigue and aerosol transport in containments, are dealt with in more detail. Some corresponding experiments are presented, together with available calculations. On the basis of analysis of experimental data and results of CFD simulations, a statement on the appropriateness of a given CFD code to the intended class of problems can be stated. This step completes the description of the existing assessment bases.

Ref. 1: “Verification and Validation of CFD Simulations”, 1999, Stern, Wilson, Coleman, Paterson (Iowa Institute of Hydraulic Research and Propulsion Research Center), report of the IIHR, (www.iihr.uiowa.edu/gothenburg2000/PDF/iihr_407.pdf).

Ref. 2: “Verification and Validation in Computational Fluid Dynamics”, 2002, Oberkampf, Trucano, Sandia National Laboratories report.

Ref. 3: “Tutorial on CFD V&V of the NPARC Alliance”, (http://www.grc.nasa.gov/WWW/wind/valid/validation.html).

Ref. 4: Shaw, R.A., Larson, T.K. & Dimenna, R.K. “Development of a phenomena identification and ranking table (PIRT) for thermal-hydraulic phenomena during a PWR LBLOCA”, NUREG/ CR-5074, EG&G Idaho, Inc., 1988.

Ref. 5: Wilson, G.E. & Boyack, B.E. “The Role of the PIRT Process in Experiments, Code Development and Code Applications Associated with Reactor Safety Analysis”, *Nuclear Engineering and Design*, **186**, 23-37 (1998).

Ref. 6: Chung, B.D. et al. “Phenomenological Identification and Ranking Tabulation for APR 1400 Direct Vessel Injection Line Break”, Proc. NURETH-10, Seoul, Korea, Oct. 5-9, 2003.

Ref. 7: C.-H. Song, et al. (2006), “Development of the PIRT for the Thermal Mixing Phenomena in the IRWST of the APR1400”, Proc. 5th Korea-Japan Symposium on Nuclear Thermal Hydraulics and Safety (NTHAS5), Jeju, Korea, Nov. 26-29, 2006

4.1 Validation Tests Performed by Major CFD Code Vendors

The code vendors identified here are those who promote general-purpose CFD: namely, ANSYS-CFX, STAR-CD, FLUENT and PHOENICS, all of whom have customers in the nuclear industry area. Other organisations with specialisations in certain areas, such as the aerospace industry, are excluded from the list, though those codes written specifically for nuclear applications, though not always available for general use, are included.

Each of the vendors operates in a commercial environment, and is keenly aware of their major competitors. Consequently, such a sensitive item as validation, which might lead them into an unwelcome code-code comparison exercise, may not receive all the attention it deserves. In addition, a validation activity may have been performed at the request of a particular customer, and the results restricted, or may not be published unless successful. Nonetheless, the companies are becoming more open, and have actively participated in international projects: the active involvement of ANSYS-CFX in the EU 5th Framework Programme ECORA is such an example.

The best source of information on specific validation databases is through the respective websites:

ANSYS-CFX	www.ansys.com
STAR-CD	www.cd-adapco.com
FLUENT	www.FLUENT.com
PHOENIX	www.cham.co.uk

Here one finds documentation, access to the workshops organised by each company, and to conferences and journal articles where customers and/or staff have published validation material. The most comprehensive documentation list appears to have been put together for PHOENICS, where a list of over 950 published papers can be found (some are validation cases), a special section devoted to validation issues is included on the website, and the code has its own journal containing peer-reviewed articles.

Clearly, the list of validation documents is too long to be written here, but evidence of its existence does confirm that commercial CFD has a well-founded technology base. It should be noted, however, that even for codes explicitly written for the nuclear community normally include basic (often academic) validation cases, just like those codes from the commercial area. A survey of validation tests has been put together by Freitas (Ref. 1).

Ref. 1: C.J. Freitas "Perspective - Selected benchmarks from commercial CFD codes" J. Fluids Engg. 117, 208.

GASFLOW

The GASFLOW code, which has been developed as a cooperation between Los Alamos National Laboratory (LANL) and Forschungszentrum Karlsruhe (FZK), is a 3D fluid dynamics field code used to analyse flow phenomena such as circulation patterns, stratification, hydrogen distribution, combustion and flame propagation, local condensation and evaporation phenomena, and aerosol entrainment, transport and deposition in reactor containments. GASFLOW is a finite-volume code, and based on robust numerical techniques for solving the compressible Navier-Stokes equations in Cartesian or cylindrical geometries. A semi-implicit solver is employed to allow large time steps. The code can model geometrically complex facilities with multiple compartments and internal structures, and has transport equations for multiple gas species, liquid water droplets, and total fluid internal energy. A built-in library contains the properties of 23 gas species and liquid water. GASFLOW can simulate the effects of two-phase dynamics with the homogeneous equilibrium model, two-phase heat transfer to and from walls and internal structures, catalytic hydrogen recombination and combustion processes, and fluid turbulence.

Ref. 1: J.R. Travis, J.W. Spore, P. Royl, K.L. Lam, T.L. Wilson, C. Müller, G.A. Necker, B.D. Nichols, R. Redlinger, "GASFLOW: A Computational Fluid Dynamics Code for Gases, Aerosols, and Combustion", Vol. I, Theory and Computational Model, Reports FZKA- 5994, LA-13357-M (1998).

Ref 2: J.W. Spore, J.R. Travis, P. Royl, K.L. Lam, T.L. Wilson, C. Müller, G.A. Necker, B.D. Nichols, "GASFLOW: A Computational Fluid Dynamics Code for Gases, Aerosols, and Combustion", Vol. II, User's Manual, Reports FZKA-5994, LA-13357-M (1998).

STAR-CD

Some elements relevant of the STAR-CD validation process are listed here: they derive from Workshop or University researches and are not nuclear oriented. CD Adapco, the company who market STAR-CD in Europe, is compiling a much more comprehensive validation list (including testing of turbulence models, heat transfer, multiphase flows, combustion, etc.), but the information is mainly derived from industrial cases, which are confidential. Consequently, it will not be readily available.

Lid-Driven Cavity Flow

The problem is characterised by its elliptic and non-linear nature: numerical diffusion is tested. This study is concentrated on using the test case to compare the performance of the code with different types of mesh. Three types of mesh are used in this calculation, namely hexahedral cells, tetrahedral cells and polyhedral (trimmed) cells.

Two-Dimensional Single Hill Flow

This is one of the two test cases prepared for the ERCOFTAC Workshop on Databases and Testing of Calculation Methods for Turbulent Flows (organised as part of the 4th ERCOFTAC/IAHR Workshop on Refined Flow Modelling. Experimental data have been provided, and the main objective of the exercise was to demonstrate the accuracy of prediction attainable. This study is concerned with the turbulent flow past a surface mounted obstacle in a channel.

Supersonic Flow Over a Flat Plate

This example concerns the development of the turbulent boundary layer on a two-dimensional wedge. The cross-sectional geometry of the wedge is an elongated trapezium with the top and bottom surfaces parallel. The leading edge is the intersection between the wedge's front and top surfaces, and the inclined angle between them is 6.7^{o}. The rear end of the wedge is vertical. Measuring from the tip of the leading edge to the trailing edge, the length of the wedge is 0.914 m. In the parallel part of the wedge, the thickness is 0.033 m.

During wind-tunnel tests, the flat surface of the wedge was kept parallel to the flow direction and hence at zero pressure gradient. The model was placed in the centre of the working section and the flow was considered to be two-dimensional. The wedge was not actively cooled, but was allowed to reach equilibrium temperature. Based on free-stream flow conditions of air, the Reynolds number was 15 350 000.

Turbulent Flow Over a Surface-Mounted Rib

This study is concerned with turbulent flow past a surface-mounted obstacle in a channel. The obstacle, representing a fence or rib, spans the whole width of the channel. Tests were performed in air at 20°C and over a range of flow velocities. Based on the mean inlet velocity and obstacle height, the Reynolds numbers ranged between 1500 to 3000.

Turbulent Vortex-Shedding around a Square Cross-Section Cylinder

This study is concerned with turbulent flow past a square-section cylinder, which exhibits natural periodic shedding of vortices. The experimental measurements were made by Durao et al. and the experimental configuration comprised a square cross-section cylinder spanning the whole width of a rectangular cross-section channel. According to their findings, the width of the test section was sufficiently large for the flow to be assumed two-dimensional at the central plane. Based on the mean flow velocity of water at inlet and on the height of the square, the Reynolds number was 14 000.

One-Dimension SOD's Shock Tube

A shock tube is simply a tube that is divided by a membrane or diaphragm into two chambers at different pressures. When the membrane is suddenly removed (broken), a wave motion is set up. This problem is characterised by the interface between the low and high-pressure chambers. The contact face, as it is known, marks the boundary between the fluids that were initially on either side of the diaphragm.

The main purpose of this validation case is to demonstrate the use of the gradient-based second order accurate differencing scheme (MARS) and the second-order temporal discretisation scheme in capturing the wave structures and motions.

Friction Factor of Fully Developed Turbulent Pipe Flow

The case of turbulent flow through pipes has been investigated thoroughly in the past, and a large amount of experimental data is available in the open literature. Because of its wide range of applications, it is also important for any CFD code to predict friction values that are comparable to those obtained from experiments.

TRIO-U (Version V1.4.4)

Non-nuclear specific test cases used as a validation database are listed here.

Laminar flow (for incompressible, Boussinesq and low Mach number regimes)

Basic tests for convection, diffusion and coupled problems:

2D Poiseuille flow; 2D axi-Poiseuille; 3D Poiseuille; 2D and 3D Taylor-Green vortices; 2D axi-symmetric pipe flow, with and without conjugate heat transfer; boundary layer on a vertical plate; flow past a 2D circular cylinder (Re=100); oscillating flow in non-symmetrically heated cavity; square box with a moving wall.

Turbulent flow (incompressible, Boussinesq and low Mach number regimes)

a) Mixing length model:

   flow in a turbulent periodic channel; flow in a turbulent periodic pipe.

b) k-epsilon model:

   2D axisymmetric pipe flow, with and without varying sections; 2D Hill flow; heated square box with unsteady thermal stratification with air inlet and outlet; differentially heated square box; S-shaped channel; flow around a single cube and around buildings (from the EEC TRAPPOS project).

c) LES modelling / RANS-LES hybrid model:

   freely decaying homogeneous isotropic turbulence; isothermal turbulent periodic channel/pipe flow with and without wall functions; differentially heated channel flow with and without wall functions, and with and without solid wall coupling; vertical impinging jet; flow around circular or square cylinders (from ERCOFTAC database); LES on specific nuclear applications.

Porous medium

a) Air flow through a particle bed; air flow in a storage room with axial arrays of heating tubes; Blasius flow with regular loss of pressure; Blasius flow with mixed open medium and porous medium.

Radiation module

a) 2D and 3D square cavity with 2 facing walls at imposed temperature and 2 facing perfectly reflecting walls; 2D and 3D axisymmetric cylinders; 2D and 3D square cavity filled with steam (for radiation in absorbing media).

Nuclear specific test cases

Some comparisons between experiments and CFD results have been performed. These include data from the ROCOM 1/5th scale reactor of FZR (Forschungszentrum Rossendorf), from the ISP-43, from tee-junction configurations, from experiments involving temperature transport, and from dilution in complex geometries.

SATURNE (Version 1.1)/NEPTUNE_CFD

Listed below are elements of the validation matrices of the EDF in-house code SATURNE, with both nuclear and non-nuclear items included. Much of the single-phase part of the coding was later incorporated in the NEPTUNE_CFD code.

1. Flow around an isolated cylinder: laminar, unsteady, isothermal regime
2. Flow in a 2D square cavity with moving wall: laminar, steady, isothermal regime
3. Taylor vortices: laminar, unsteady, isothermal regime
4. Plane channel flow: laminar and turbulent, steady, isothermal regimes
5. 2D Flow over a hill: turbulent steady, isothermal regime
6. 2D flow in a 2D arrays of tubes: turbulent, steady and unsteady, isothermal regime
7. Flow in a 2D channel with inclined pressure drop: laminar, steady, isothermal regime
8. Freely decaying homogeneous isotropic turbulence: turbulent, unsteady, isothermal regime

9. 3D flow in a cylindrical 180° curved pipe: steady, turbulent, isothermal regime
10. 3D flow around a car shape: steady, turbulent, isothermal regime
11. Natural convection in a 2D closed box with vertical heated walls: steady, turbulent, natural-convection regime
12. Mixed convection in a 2D cavity with air inflow and heating: steady, turbulent, mixed-convection regime.
13. Mixed convection in a 2D cavity with heated floor and air circulation heating: steady, turbulent, mixed-convection regime.
14. 2D axisymmetric jet impingement on a heated wall: steady, turbulent, forced-convection regime.
15. 2D axisymmetric jet of sodium: steady, turbulent regime with thermal transfer
16. Thermal stratification in a hot duct with cold water injection: steady, turbulent, stratified regime
17. Injection (at 45°) of a mixture of gases in a pure gas: steady, turbulent, multi-species flow
18. 2D channel with thick heated walls: steady, turbulent flow with thermal coupling
19. Premixed combustion: steady reactive turbulent flow
20. Diffusion flame: steady, reactive, turbulent flow
21. Pulverised coal furnace: steady, turbulent, reactive flow with radiation heat transfer
22. Two-phase gas/particle flow along a vertical plate: steady, turbulent flow with Lagrangian transport
23. Two-phase gas/particle flow in a vertical cylindrical duct: steady, turbulent flow with Lagrangian transport
24. Industrial tee-junction: steady, turbulent flow
25. Industrial cold water injection in hot water duct: unsteady, turbulent flow with heat transfer
26. Simple tests of functionalities of practical interest (parallelism, periodicity, restart…)
27. Analytical case of radiative transfer in a closed cavity: steady, radiation heat transfer

Cast3M (including TONUS)

Listed below are elements of the validation matrices of two CEA in-house codes; both nuclear and non-nuclear items are included.

Test of scalar equation transport (academic test cases)

a. Convection: 2D rotational transport flow
b. Convection-diffusion: 2D Smith-Hutton flow
c. Non-linear conservation law: 2D Burgers equation
d. Diffusive transport: 2D and 3D heat equation

Radiation heat transfer

a. Transparent media: square cavity, wedge, co-axial cylinders, co-centric spheres, cube
b. Radiation and conduction: air-filled cylinder

c. Absorbing media: absorbing gas in a sphere
d. Radiation and natural convection in absorbing media: 2D square cavity

Single-Component Flow

a. Incompressible
    i. Lid-driven cavity
    ii. Blasius flat plate
    iii. Backward-facing step
b. Boussinesq
    iv. Natural convection in zero Prandtl fluid
    v. Rayleigh-Marangoni convection
    vi. Vahl Davis differentially heated cavity
c. Low Mach Number
    vii. Differentially heated cavity with large temperature differences
    viii. Pressurisation
d. Compressible Flows
    ix. 2D Laval-type nozzles or channel flow; 1D SOD shock tube; 1D double rarefaction wave; shock collisions; moving or steady contact waves; moving or steady shock waves; 1D blast wave; 2D shock reflection; 2D inviscid shear layer; 2D jet interaction; odd-even decoupling; “Carbuncle Test Case”; double Mach reflection; forward-facing step; shock diffraction over 90° corner.
e. Multi-Component Flows
    x. Low Mach and compressible approaches; shear layer; non-reactive shock tube; reactive shock tube.
f. Turbulence Modelling
    xi. Incompressible k-eps: grid turbulence; fully-developed channel flow; turbulent natural convection in a square cavity
    xii. LES on specific experiments
    xiii. k-eps and Mixing-Length model for low Mach number NS Equations with condensation
    xiv. k-eps for low Mach number reactive flows (EBU modelling)
g. Containment
    xv. MISTRA tests
    xvi. Wall condensation experiment
    xvii. Condensation + convection + conduction in axisymmetric and 3D geometries, with and without He
    xviii. Flow in 3D compartmented geometries
    xix. Spray dynamics, with convective heat transfer
    xx. Droplet heat and mass transfer

xxi. Spray experiments

xxii. H2 detonation in 1D, 2D and 3D geometries

xxiii. Fast and slow H2 deflagrations

xxiv. LP models with H2 recombiner, with stratification and distribution, with wall condensation

xxv. Air/steam leaks in idealised and concrete cracks

h. "GCR" Specific Models

xxvi. Conduction, radiation, convection in complex geometries

xxvii. Turbine blade deblading

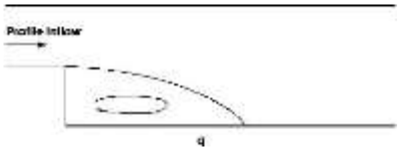
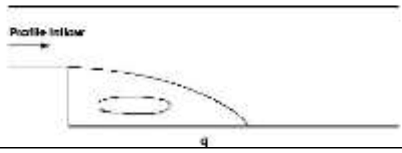
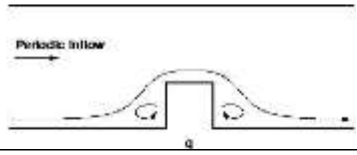
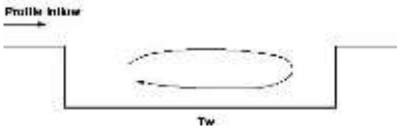
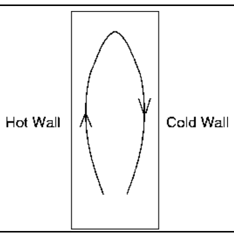
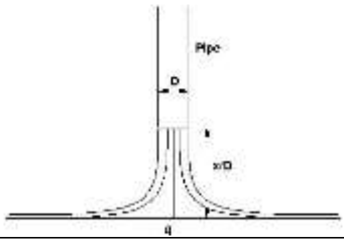
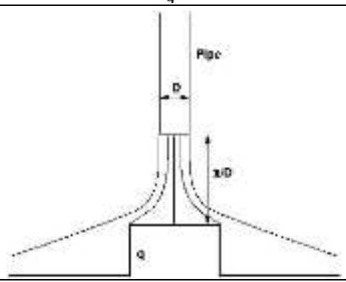
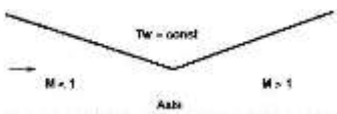
ANSYS (ANSYS-CFX)

Heat transfer predictions from the two codes ANSYS-TASCflow and ANSYS-CFX are comprehensively covered in the document cited below. All situations analysed were for turbulent flow conditions. Three two-equation, eddy-viscosity turbulence models were analysed in the context of 9 test cases, illustrated in the accompanying table The test cases are idealised, academic standards, but nonetheless of relevance to NRS issues, since many such situations (though not idealised) will occur in NRS applications. It is estimated that less than 1% of all industrial applications of CFD target the prediction of heat transfer to and from solid walls.

It was found that the often reported poor performance of eddy viscosity models could be attributed to the application of low-Re near wall treatments, and not so much on the underlying turbulence model. It is generally known that k-ε approaches overpredict heat transfer rates in regions of adverse pressure gradient, and at flow-attachment points. The k-ω model has better heat transfer characteristics in near-wall regions, but is sensitive to the free-stream values of ω outside the wall boundary layer. The sensitivity often extends to the specification of inlet values. The SST (Shear-Stress Transport) model is an attempt to take advantage of the favourable characteristics of both models by combining a k-ω treatment near the wall and a k-ε description in the far field. This model performed the best in all 9 test cases, and results compared well with more complex four-equation model v2f, developed at Stanford. On the basis of this benchmark exercise, it was demonstrated that the ANSYS-CFX software is capable of performing heat transfer simulations for industrial flows. The experience gained from this exercise endorses the statement that CFD is a "tried-and-tested" technology, and this has immediate benefits for NRS applications.

Overall, validation is a key component of the ANSYS-CFX software strategy, which is reflected in the vendor's participation in international benchmarking activities, such as those organised within EU Framework Programmes (ASTAR, ECORA) and ERCOFTAC.

Ref. 1: W. Vieser, T. Esch, F. Menter "Heat Transfer Predictions using Advanced Two-Equation Turbulence Models", ANSYS-CFX Technical Memorandum, ANSYS-CFX-VAL10/0602, AEA Technology, June 2002, florian.menter@ansys.com.

	Experiment	Mach N° & Fluid Properties	Flow Type	Items of Interest
Backward Facing Step		Ideal Gas	Plane, 2D	Flow separation, reattachment and re-developing flow (Vogel & Eaton, 1985)
Pipe Expansion	Rotation axis 	Ideal Gas	Axi-symmetric	Flow separation, reattachment and re-developing flow (Baughn et al., 1984)
2D-Rib		Ideal Gas	Plane, 2D	Periodic flow over a surface mounted rib (Nicklin, 1998)
Driven Cavity		Ideal gas	Plane, 2D	Driven cavity flow, (Metzger et al., 1989)
Natural Convection		Ideal gas	Plane, 2D	Buoyancy, heat transfer (Betts & Bokhari, 2000)
Impinging Jet		Ideal gas	Axi-symmetric	Stagnation flow, (Craft et al., 1983; Yan et al., 1992)
Impinging Jet on a Pedestal		Ideal Gas	Axi-symmetric	Stagnation flow, (Baughn et al., 1993; Mesbah, 1996)
Subsonic and Supersonic Nozzle Flow		0.2 – 2.5, air-methane mixture, ideal gas	Axi-symmetric	Cooled turbulent boundary layer under the influence of large pressure gradients (Back et al., 1964)

Ref. 1: Back, L.H., Massier, P.F. and Gier, H.L., 1964, "Convective Heat Transfer in a Convergent-Divergent Nozzle", Int. J. Heat Mass Transfer, Vol. 7, pp. 549 – 568

Ref. 2: Baughn, J.W., Hoffmann, M.A., Takahashi, R.K. and Launder, B.E., 1984, "Local Heat Transfer Downstream of an Abrupt Expansion in a Circular Channel With Constant Wall Heat Flux", Vol. 106, Journal of Heat Transfer, pp. 789 – 796.

Ref. 3: Baughn, J. W., Mesbah, M., and Yan, X., 1993, "Measurements of local heat transfer for an impinging jet on a cylindrical pedestal", ASME HTD-Vol 239, pp. 57-62

Ref. 4: Betts, P. L., and Bokhari, I. H., 2000, "Experiments on turbulent natural convection in an enclosed tall cavity", Int. J. Heat & Fluid Flow, 21, pp. 675-683

Ref. 5: Craft, T. J., Graham, L. J. W., and Launder, B. E., 1993, "Impinging jet studies for turbulence model assessment – II. An examination of the performance of four turbulence models", Int. J. Heat Mass Transfer. 36(10), pp. 2685-2697

Ref. 6: Mesbah, M., 1996, "An experimental study of local heat transfer to an impinging jet on non-flat surfaces: a cylindrical pedestal and a hemispherically concave surface", PhD Thesis, University of California, Davis.

Ref. 7: Metzger, D. E., Bunker, R. S., and Chyu, R. K., 1989, "Cavity Heat Transfer on a Transverse Grooved Wall in a Narrow Channel", J. Heat Transfer, 111, pp. 73-79

Ref. 8: Nicklin, G. J. E., 1998, "Augmented heat transfer in a square channel with asymmetrical turbulence production", Final year project report, Dept. of Mech. Eng., UMIST, Manchester

Ref. 9: Vogel, J.C. and Eaton, J.K., 1985, "Combined Heat Transfer and Fluid Dynamic Measurements Downstream of a Backward-Facing Step", Vol. 107, Journal of Heat Transfer, pp. 922 – 929.

Ref. 10: Yan, X., Baughn, J. W., and Mesbah, M., 1992, "The effect of Reynolds number on the heat transfer distribution from a flat plate to an impinging jet", ASME HTD-Vol 226, pp. 1-7.

FLUENT

A generally available validation database for FLUENT does not currently exists. There are instead three levels of validation reports. The most public are journal publications of validation exercises. Since 1990, more than 100 references have accrued citing validation activities; of these 6 were related to NRS applications. At a second, and more restrictive level, FLUENT provides licensed code users (for Universities only the primary holder of the site license) with online access to nineteen validation reports. Titles of the reports are:

Flow in a Rotating Cavity

Natural Convection in an Annulus

Laminar Flow Around a Circular Cylinder

Flow in a 90 Planar Tee-Junction

Flows in Driven Cavities

Periodic Flow in a Wavy Channel

Heat Transfer in a Pipe Expansion

Propane Jet in a Coaxial Air Flow

Non-Premixed Hydrogen/Air Flame

300 kW BERL Combustor

Flow through an Engine Inlet Valve

Turbulent Flow in a Transition Duct

Solid Body Rotation with Central Air Injection

Transonic Flow Over a RAE 2822 Airfoil

Mid-Span Flow Over a Goldman Stator Blade

Compressible Turbulent Mixing Layer

Scramjet Outflow

Turbulent Bubbly Flows

Adiabatic Compression and Expansion Inside an Idealised 2D In-Cylinder Engine.

The third, and more detailed, set of validation reports exists internal to FLUENT. The tests are applied during development of new code versions, but most are proprietary, and details of this validation set are not available externally.

Ref. 1: F. Lin, B. T. Smith, G. E. Hecker, P. N. Hopping, "Innovative 3-D numerical simulation of thermal discharge from Browns Ferry multiport diffusers", Proc. 2003 International Joint Power Generation Conference, Atlanta, GA, June 16-19 2003, p 101-110.

Ref. 2: R. M. Underhill, S. J. Rees, H. Fowler, "A novel approach to coupling the fluid and structural analysis of a boiler nozzle", Nuclear Energy, 42(2), 95-103 (2003).

Ref. 3: T.-S. Kwon, C.-R. Choi, C.-H. Song, "Three-dimensional analysis of flow characteristics on the reactor vessel downcomer during the late reflood phase of a postulated LBLOCA", Nucl. Eng. Des., 226(3), 255-265 (2003).

4.2 ERCOFTAC

The European Research Community on Flow, Turbulence And Combustion (ERCOFTAC) is an association of research, educational and industrial groups with main objectives to promote joint efforts, centres and industrial application of research, and the creation of Special Interest Groups (SIGs).

A large number of SIGs have been formed, and one is the ERCOFTAC Database Interest Group (DBig), with the objective to coordinate, maintain and promote the creation of suitable databases derived from experimental, DNS, LES, CFD, PIV and flow visualisation specialists.

This data base, started in 1995, and administrated by UMIST Mechanical Engineering CFD group, contains experimental as well as existing numerical data (collected through Workshops) relative to both academic and more applied applications. The database is actively maintained by UMIST staff, and is currently undergoing a restructuring and expansion to include, amongst other things, more details of the test cases, computational results, and results and conclusions drawn from the ERCOFTAC Workshops on Refined Turbulence Modelling. Each case contains at least a brief description, some data to download, and references to published work. Some cases contain significantly more information than this.

ERCOFTAC databases can be found for four basic sources:

- **Classic Data Base**, which is open to the public (but registration is needed when downloading data). Documented are 83 cases, either containing experimental data, or with DNS/LES data available. Some of the cases could be used also in NRS applications, such as flow in curved channels, mixing layers, and flows through tube bundles.

- **Experimental Distributed Data Base** is under development and aims to collect web-accessible experimental datasets that are of potential interest to the wider community of flow, turbulence and combustion researchers, engineers and designers. Currently of special interest from the point of view of nuclear reactor safety are the Barton Smith (Utah State University) experimental data, since they contain pressure drop and velocity field measurements for flow through an array of cylinders. These mimic a Next Generation Nuclear Plant lower plenum, with measurements of velocity and turbulence for flow along fuel rods separated by grid spacers, performed within the project "Advanced computational thermal fluid physics (CTFP) and its assessment for light water reactors and supercritical reactors'. Experimental data may be downloaded in the form of ASCII files. Animations are available, together with reports describing the experimental arrangements.
- **DNS/LES Distributed Data Base** is also under development and contains links to several papers describing applications of DNS and LES, with detailed experimental and computational data. There is also a link to the DNS data base of the Turbulence and Heat Transfer Laboratory, University of Tokyo. The DNS data base is openly available, but some other links within this page require user ID and password. The data are related to basic problems of turbulence and do not have direct application to engineering analyses.
- **Distributed Flow Visualisation Library** is currently available in French only; a version in English is under construction. The library contains at present almost 300 items, including authors, title, keywords and abstracts, but loading them requires postal delivery of a CD ROM. Visualisations from both experiments and numerical analyses are included, some of them (e.g. visualisation of liquid-gas bubbly flow, No. 40) could be interesting to developers of two-phase flow models. Information on flow patterns in various geometries and flow regimes can also help in assessment of CFD simulations.

Current and past test cases of three Special Interest Groups (SIG's), namely Turbulence Modelling SIG, Transition Modelling in Turbomachinery SIG, and Large Eddy Simulation SIG can be found via the referenced links, as well as links to worldwide fluid dynamics data bases. Unfortunately, for several links, the web sites probably do not now exist.

www.ercoftac.org

Classic Data Base:
http://cfd.me.umist.ac.uk/ercoftac/

Experimental Distributed Data Base:
http://ercoftac.mech.surrey.ac.uk/exp/homepage.html, http://www.mae.usu.edu/faculty/bsmith/data.html,
http://www.mae.usu.edu/faculty/bsmith/EFDL/array/Array.html,
http://www.mae.usu.edu/faculty/bsmith/EFDL/KNERI/KNERI.html

DNS/LES Distributed Data Base:
http://ercoftac.mech.surrey.ac.uk/dns/homepage.html,
http://www.thtlab.t.u-tokyo.ac.jp/,

Distributed Flow Visualisation Library:
http://ercoftac.mech.surrey.ac.uk/flovis/homepage.html

Special Interest Groups:
http://tmdb.ws.tn.tudelft.nl/,
http://ercoftac.mech.surrey.ac.uk/transition/homepage.html,
http://ercoftac.mech.surrey.ac.uk/LESig/homepage.html

Worldwide Data Base:
http://ercoftac.mech.surrey.ac.uk/links/data.html

4.3 QNET-CFD Knowledge Base

QNET-CFD is "A Thematic Network for Quality and Trust in the Industrial Application of Computational Fluid Dynamics", partly funded by the EU. Four years were spent in assembling and collating knowledge and know-how across a range of CFD applications. The resulting knowledge base will be launched shortly into the public domain under the stewardship of ERCOFTAC, but limited access is possible now.

The knowledge base is hierarchically structured around the notions of *Application Areas*, *Application Challenges* (realistic test cases which can be used in assessment of CFD for a given Application Area), and *Underlying Flow Regimes* (generic, well studied test cases capturing important elements of the key flow physics encountered in one or more Application Challenges). Each Application Challenge and Underlying Flow Regime features best practice advice providing guidance on model set-up decisions and the interpretation of results.

At present, the following Application Areas are included:

- External Aerodynamics
- Combustion and Heat Transfer
- Chemical and Process, Thermal Hydraulics and Nuclear Safety
- Civil Construction and HVAC
- Environmental Flow
- Turbomachinery Internal Flow.

In the Chemical and Process, Thermal Hydraulics and Nuclear Safety Application Area, the following Application Challenges are included:

- **Buoyancy-opposed wall jet** (contributed by Magnox Electric, UK); a two-dimensional buoyancy-opposed plane wall jet penetrating into a slowly moving, counter-current uniform flow. Experimental study of this flow has been performed at the University of Manchester (UMIST) using a water rig. Particle Image Velocimetry (PIV) and Laser Doppler Anemometry (LDA) systems were used to study the mean flow and turbulent fields. Laser light sheet flow visualisation and PIV were used to obtain pictures of the instantaneous flow structure. Detailed measurements of local mean velocity, turbulence and temperature were then made using an LDA system incorporating a fibre optic probe and transversable rake of thermocouples. Computations have been performed at UMIST using the two-dimensional finite-volume TEAM code. Four models of turbulence based on RANS and a LES model have been considered. The jet-spreading rate (distance from the wall where the mean velocity becomes half the local maximum velocity), and the jet penetration depth were chosen to assess the quality of the numerical simulations.
- **Induced flow in a T-junction** (contributed by the EDF R&D Division, Chatou, F); a high-Reynolds number flow is maintained in the main pipe while very small incoming mass flow rates are imposed in the auxiliary pipe. Description of the swirl flow in the auxiliary leg should be well predicted. Experiments have been performed at Chatou, and two RANS turbulence models (k-epsilon, and RSM) have been used in the calculations. The height of the swirl is the main parameter to assess the quality of calculations.
- **Cyclone separator** (contributed by FLUENT Europe Ltd) No details yet available.

- **Buoyant gas air-mixing** (contributed by British Nuclear Fuels, BNFL, UK); the mixing of buoyant gas (helium or hydrogen) with air in a vessel. The mole fraction of hydrogen or helium measured at various points in the geometry is the assessment parameter.
- **Mixed convection in a reactor** (contributed by CEA/DMT Saclay, F); distribution of steam and /or hydrogen in containment during an accident with break in the reactor coolant system. Experiment D30 of the MISTRA experimental series, which focused on validation of turbulence and condensation models, was selected. CFD simulation with the CEA code TONUS is presented. The objective is to predict correctly condensation rates and gas distribution in the cylindrical containment. The effect of turbulence on the mixing of scalars (temperature, concentrations), and on pressure and condensation rates are the key parameters.
- **Spray evaporation in turbulent flow** (contributed by Martin-Luther-Universität, Halle-Wittenberg, D); spray evaporation in a heated turbulent air stream was studied experimentally with isopropyl-alcohol used as liquid. Different flow conditions (flow rate, air temperature, liquid flow rate) were studied in a pipe expansion (with expansion ratio of three). Heated air entered through an annulus, and there was a hollow cone-spray nozzle mounted at the centre. Phase-Doppler anemometry (PDA) was applied to obtain the spatial change of the droplet size spectrum in the flow field and to measure droplet size-velocity correlations. Profiles of droplet mean velocities, velocity fluctuations, and droplet mean diameters were then obtained by averaging over all droplet size classes, and profiles of droplet mass flux, enabling determination of global evaporation rates, were also determined. Velocity profiles of both phases along the test section, including mean velocities for the axial and radial components as well as the associated rms-values, are the assessment parameters. Additionally, profiles of droplet mean diameters and droplet mass flux can be used, together with the liquid mass flow along the test section, enabling the global evaporation rate to be determined.
- **Combining/dividing flow in Y junction** (contributed by Rolls-Royce Marine Power, Engineering & Technology Division) No details yet available.
- **Downward flow in a heated annulus** (contributed by British Energy, UK); turbulent downward flow in an annulus with a uniformly heated core and an adiabatic outer casing was tested with the aim of evaluating the influence of buoyancy on mixed-convection flow, heat transfer and turbulence. The Reynolds number of the flows ranges from 1000 to 6000, and the Grashof number (based on heat flux) ranges from 1.1×10^8 to 1.4×10^9. The experimental data collected on the experimental rig in the Nuclear Engineering Department, School of Engineering, University of Manchester are temperatures, velocity and turbulence. A representative set of CFD calculations have been undertaken at UMIST using the k-epsilon turbulence model, but with three approaches to the modelling of near-wall turbulence. The variation of Nusselt number on the heated core is the assessment parameter.

For each Application Challenge, its description, test data, CFD simulations, evaluation, best practice advice, and related underlying flow regimes should all be available. At present, user ID and password are required.

Ref. 1: http://eddie.mech.surrey.ac.uk/homepage.htm

4.4 MARNET

These are Best Practices Guidelines for Marine Applications of CFD, and were prepared by WS Atkins Consultants. The general ERCOFTAC document is taken as a starting point, and specific advice on the application of CFD methods within the marine industry are provided.

Ref. 1: WS Atkins Consultants, "Best Practices Guidelines for Marine Applications of CFD," MARNET-CFD Report, 2002.

Ref. 2: https://pronet.wsatkins.co.uk/marnet/

4.5 FLOWNET

The FLOWNET initiative is intended to provide the scientific and industrial communities with a code validation tool for flow modelling and computational/experimental methods. By means of network databases, multi-disciplinary knowledge is cross-fertilised and archived. Providing a share of technical complements to scientists and engineers, the network enhances quality and trust in pre-industrial processes. The ultimate goal of the network is to bring together academic and industrial node partners in a dynamically open forum to evaluate continuously the quality and performance of CFD software for improving complex design in industry from the viewpoint of accuracy and efficiency. The FLOWNET project provides data once specific authorisation has been provided; the main orientation is the aerodynamics community (http://dataserv.inria.fr/sinus/flownet/links/index.php3).

4.6 NPARC Alliance Data Base

The NPARC Alliance for CFD Verification & Validation provides a tutorial, as well as available measurements and data for CFD cases, chiefly orientated towards the aerodynamics community. The data archive of NASA also provides suitable data for CFD applications, while there is also a link to an archive of the high-quality validation data listed below.

- Incompressible, turbulent 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;
- ONERA M6 wing;
- 2D axisymmetric boat tail nozzle;
- 3D boat tail nozzle
- Hydrogen-air combustion in a channel;
- Dual-stream mixing;
- Laminar flow over a circular cylinder.

All validation cases include a full flow description, comparison data and references.

Measurements and Data:
http://www.grc.nasa.gov/WWW/wind/valid/tutorial/tutorial.html

NASA Archive:
http://www.nas.nasa.gov/Software/DataSets

Validation Data:
http://www.grc.nasa.gov/WWW/wind/valid/

4.7 AIAA

The American Institute of Aeronautics and Astronautics, or AIAA, is a 65-year-old “professional society for aerospace professionals in the United States”. Its purpose it to “advance the arts, sciences, and technology of aeronautics and astronautics, and to promote the professionalism of those engaged in these pursuits”. For example, there is a link up with the QNET-CFD activity. The society participates to the definition of standards for CFD in its “Verification and Validation Guide”.

Web sites related to AIAA activities propose lists of references (papers, books, author coordinates) related to CFD verification and validation and various links with other web sites gathering information of aeronautical interest. Some of these links may provide valuable information for CFD validation, though this would have to be sifted for information of interest to the NRS community.

Base address:
http://www.aiaa.org

CFD V&V:
http://www.aiaa.org/publications/database.html
http://www.icase.edu/docs/library/itrs.html

4.8 Vattenfall Database

The Plane Wall Jet (UFR3-10)

Detailed three-component turbulence measurements in a wall jet down to $y^+<2$ are reported. The experimental technique was a combination of light collection in 90° side-scatter, and the use of optics with probe volumes of small diameters. A complete k-profile was obtained, and turbulence statistics up to fourth order are presented for all three velocity components. Comparing the wall jet to the flat plate boundary layer, one finds that the turbulence structure in the near-wall region is qualitatively very similar, but that the actual values of the quantities (in conventional inner scaling) are higher for the wall jet.

Draft Tube (TA6-07) for a Kaplan Turbine

Data have been made available from measurements taken using LDV in a model turbine (scale 1:11) at Vattenfall Utveckling, Älvkarleby, Sweden for an ERCOFTAC/IAHR sponsored Workshop: Turbine 99 - Workshop on Draft Tube Flow, held at Porjus, Sweden on 20-23 June, 1999. The basic challenge for calculations submitted to the Workshop was to predict technically relevant quantities from measured data at the inlet and outlet of the draft tube. This involved head loss coefficients, pressure distributions and the positions of separated flow regions. A substantial amount of additional experimental data was made available to the participants at the meeting, involving velocity fields at several internal points, boundary layer profiles at selected points, and visual observations (with laser-induced fluorescence) of swirl and recirculation zones. Proceedings of the Workshop are available on the web at http://www.sirius.luth.se/strl/Turbine-99/index.htm, and the benchmark is also referenced in QNET-CFD.

Ref. 1: Eriksson J; Karlsson R; Persson J “An Experimental Study of a Two-Dimensional Plane Turbulent Wall Jet”, *Exp. Fluids,* **25**, 50-60 (1998).

Ref. 2: Andersson, U., Karlsson, R., "Quality aspects of the Turbine-99 experiments", in Proceedings of Turbine-99 – Workshop on draft tube flow in Porjus, Sweden, 20-23 June 1999.

Ref. 3: The QNET-CFD Network Newsletter, A Thematic Network For Quality and Trust, Volume 2, No. 3 – December 2003.

4.9 Existing CFD Databases from NEA/CSNI and Other Sources

	Source	Reference
1	State-of-the Art Report (SOAR) on Containment Thermal-Hydraulics and Hydrogen Distribution	NEA/CSNI/R(1999)16
2	SOAR on Flame Acceleration and Deflagration-to-Detonation Transition in Nuclear Safety	NEA/CSNI/R(2000)7
3	Summary and Conclusions of the May 1996 (Winnipeg) Workshop on the Implementation of Hydrogen Mitigation Techniques	NEA/CSNI/R(1996)8
4	Proceedings of the 1996 (Annapolis) Workshop on Transient Thermal-Hydraulic and Neutronic Code Requirements	NEA/CSNI/R(1997)4
5	Proceedings of the April 2000 (Barcelona) Workshop on Advanced Thermal-Hydraulic and Neutronic Codes - Current and Future Applications (Volumes 1 and 2)	NEA/CSNI/R(2001)2
6	Summary and Conclusions of the April 2000 (Barcelona) Workshop on Advanced Thermal-Hydraulic and Neutronic Codes - Current and Future Applications	NEA/CSNI/R(2001)9
7	Proceedings of the May 2002 (Aix-en-Provence) Exploratory Meeting of Experts to Define an Action Plan on the Application of CFD to NRS Problems	NEA/CSNI/R(2002)16
8	Proceedings of the November 2002 (Pisa) IAEA/NEA Technical Meeting on the Use of Computational Fluid Dynamics Codes for Safety Analysis of Reactor Systems, Including Containment	NEA/CSNI/R(2003)
9	Severe Accident Research and Management in Nordic Countries -- A Status Report, May 2000	NKS-71 (2002)
10	NKS Recriticality Calculation with GENFLO Code for the BWR Core After Steal Explosion in the Lower Head, December 2002	NKS-83 ISBN 87-7893-140-1
11	The Marviken Full-Scale Experiments	CSNI Report No. 103
12	Analysis of Primary Loop Flows (ECORA WP2 Report)	http://domino.grs.de/ecora/ecora.nsf

4.10 Euratom Framework Programmes

ASTAR

ASTAR (Advanced Three-Dimensional Two-Phase Flow Simulation Tool) was a 5th Framework EU shared-cost action dedicated to the further development of high-resolution numerical methods, and their application to transient two-phase flow. The project explored the capabilities of using hyperbolic numerical methods – which are traditionally the province of single-phase fluid dynamics, especially in the aerospace industry – for two-phase flow simulations of relevance to nuclear reactor modelling. Several benchmark exercises were adopted as verification and assessment procedures for comparing the different modelling and numerical approaches.

It was recognised that the simulation tools currently used by the nuclear reactor community are based on elliptic solvers, and suffer from high numerical diffusion. However, many of the accident sequences being modelled with these methods involve propagation of strong parameter gradients: e.g. quench fronts, stratification, phase separation, thermal shocks, critical flow conditions, etc., and such “fronts” become smeared, unless very fine nodalisation is employed. Hyperbolic methods, on the other hand, are well suited to such propagation phenomena, and one the principal goals of the ASTAR project was to demonstrate the flow modelling capabilities and robustness of such techniques in idealised, nuclear accident situations.

ASTAR provide a forum in which separate organisations, developing in-house hyperbolic solvers, could assess their progress within a common framework. To this purpose, a set of benchmark exercises were defined to which the various participants were invited to submit sample solutions. The benchmarks were taken from the nuclear research community, and for which reliable analytical, numerical or experimental data were available. These included: phase separation in a vertical pipe, dispersed two-phase flow in a nozzle, oscillating manometer, the Ransom faucet problem, the CANON (fast depressurisation) test, boiling in a vertical channel, and LINX bubble-plume tests.

Although not all the different numerical approaches (though all hyperbolic) had reached the same level of development and testing, there was evidence coming out of the project that high-resolution, characteristic-based numerical schemes have reached a satisfactory level of maturity, and might therefore be considered as alternatives to the present elliptic-based methods for a new generation of nuclear reactor thermal-hydraulic simulation tool.

Ref. 1: H. Städtke et al. "The ASTAR Project – Status and Perspective", 10th Int. Topical Mtg. on Nuclear Reactor Thermal Hydraulics (NURETH-10), Seoul, Korea, Oct. 5-9, 2003.

Ref. 2: H. Paillere et al. "Advanced Three-Dimensional Two-Phase Flow Simulation Tools for Application to Reactor Safety (ASTAR)", FISA-2003 / EU Research in Reactor Safety, 10-13 November 2003, EC Luxembourg, http://www.cordis.lu/fp5-euratom/src/ev-fisa2003.htm.

ECORA

The overall objective of the European 5th Framework Programme ECORA wass to evaluate the capabilities of CFD software packages in relation to simulating flows in the primary system and containment of nuclear reactors. The interest in the application of CFD methods arises from the importance of three-dimensional effects, which cannot be represented by traditional one-dimensional system codes. Perspective areas of the application of detailed three-dimensional CFD calculations was identified, and recommendations for code improvements necessary for a comprehensive simulations of safety-relevant accident scenarios for future research were provided. Within the ECORA project, the experience of the twelve partners from European industry and research organisations in the field of nuclear safety was combined, applying the CFD codes ANSYS-CFX, FLUENT, SATURNE, STAR-CD and TRIO_U.

The assessment included the establishment of Best Practice Guidelines and standards regarding the use of CFD software, and evaluation of results for safety analysis. CFD quality criteria is being standardised prior to the application of different CFD software packages, and results are only accepted if the set quality criteria are satisfied. Thus, a general basis is being formed for assessing merits and weaknesses of particular models and codes on a European-wide basis. CFD simulations achieving the accepted quality level will increase confidence in the application of CFD-tools to nuclear issues.

Furthermore, a comprehensive and systematic software engineering approach for extending and customising CFD codes for nuclear safety analyses has been formulated and applied. The adaptation of CFD software for nuclear reactor flow simulations is being demonstrated by implementing enhanced two-phase flow, turbulence and energy transfer models relevant for pressurised thermal shock (PTS) studies into ANSYS-CFX, Saturne and Trio_U. An analysis of selected experiments from the UPTF and PANDA test series is being performed to validate CFD software in relation to PTS phenomena in the primary system, and severe accident management in the containment.

The selected tests with PTS relevant flow phenomena include free surfaces, stratification, turbulent mixing and jet flows. The test matrix starts with single-effect tests of increasing complexity, and ends with industrially (reactor safety) relevant demonstration cases.

Verification test cases

VER01: Gravitational oscillation of water in U-shaped tube (Ransom, 1992)

VER02: Centralised liquid sloshing in a cylindrical pool (Maschek et al., 1992)

Validation test cases

VAL01: Axisymmetric single-phase air jet in air environment, impinging on a heated flat plate (Baughn and Shimizu, 1989)

VAL02: Water jet in air environment impinging on an inclined flat plate, (Kvicinsky et al., 2002)

VAL03: Jet impingement on a free surface (Bonetto and Lahey, 1993)

VAL04: Contact condensation on stratified steam/water flow (Goldbrunner et al., 1998)

Demonstration test cases

DEM01: UPTF Test 1

DEM02: UPTF TRAM C1

The ECORA web address is http://domino.grs.de/ecora/ecora.nsf, where all project documents may be found.

Ref. 1: M. Scheuerer et al., "Evaluation of computational fluid dynamic methods for reactor safety analysis (ECORA)", Nucl. Eng. Des., **235**, 359–368 (2005).

TEMPEST

The shared-cost EU FP5 project TEMPEST focussed on resolving outstanding issues concerning the effect of light gases on the long-term LOCA response of the passive containment cooling systems for the SWR1000 and ESBWR advanced reactors. Validation of multi-dimensional codes for containment analysis was a further objective. A series of five tests in the PANDA facility at PSI, with detailed local measurements of gas species, temperature and pressure, were performed within the project. The experimental data were used for the validation of CFD containment models, and provided improved confidence in the performance of passive heat-removal systems in the presence of hydrogen. CFD codes were successfully employed for predicting stratification behaviour in the containment volumes. This included finding the cause of the tendency of system codes to overpredict containment end-pressure in the presence of light gases. Improved passive containment models for the lumped parameter codes WAVCO and SPECTRA were also validated.

The TEMPEST project was begun to settle the following issues:

1) How does mixing or stratification affect long-term containment pressure response?

2) What are the effects of hydrogen on the performance of passive containment cooling systems?

3) How to apply CFD (and CFD-like) codes for improved passive containment analysis?

A threefold approach was followed. Firstly, PANDA (PSI) and KALI (CEA, Cadarache) experiments were performed in order to provide an experimental database for the above issues. Secondly, CFD models for quantitative assessment of Building Condenser (BC) and Passive Containment Cooling (PCC) system performance were developed and validated. Thirdly, both lumped-parameter and CFD (or CFD-like) codes were then applied to assist in interpreting experimental results, with the objective of better understanding passive containment behaviour.

From the analyses performed within the TEMPEST project, it was found that stratification affects the system end-pressure in these reactors through its effect on the distribution of light gases between the Drywell and the Suppression Chamber. Lumped-parameter codes demonstrated overall satisfactory performance in passive containment analyses, but showed a tendency to overpredict system end-pressure, due to their inability to properly account for stratification. In contrast, CFD codes were shown to be able to accurately predict stratification in gas spaces and water pools, and therefore produce better end-pressure predictions. A combined system-code/CFD-code approach, in which stratification is predicted using CFD, could be considered for future analyses.

Ref. 1: V.A. Wichers et al. "Testing and Enhanced Modelling of Passive Evolutionary Systems Technology for Containment Cooling (TEMPEST)", FISA-2003/EU Research in Reactor Safety, 10-13 Nov. 2003, EC Luxembourg, http://www.cordis.lu/fp5-euratom/src/ev-fisa2003.htm.

IPSS

IPSS is an acronym for European BWR R&D Cluster for Innovative Passive Safety Systems, which was an EU FP4 project concentrating on important innovations of BWRs, such as natural convection in the reactor coolant system and passive decay-heat removal. Experiments were performed at the NOKO (FZJ, separate-effects tests) and PANDA (PSI, integral tests) facilities, and post-test analyses performed with the lumped-parameter/system codes ATHLET, APROS, COCOSYS, MELCOR, RELAP5, TRAC, the containment code GOTHIC, and the CFD codes ANSYS-CFX-4 and PHOENICS.

Though it was demonstrated that traditional lumped-parameter and system codes were capable of reproducing the experimental results, it became evident that CFD codes have to be used to a greater extent than was envisaged at the start of the project. However, it was noted that the validation of these codes for commercial reactor applications was not yet satisfactory, due to the limited amount of relevant experimental data. Nonetheless, the continuing development of CFD codes, and the increasing capacity and speed of computers, the project recognised the usefulness of applying the codes to the analysis of thermal-hydraulic phenomena in real reactors in the future. It was also recommended to continue the study of flow and temperature fields in large water pools and in the containment, and perform further experiments with improved instrumentation (increase in number and sometimes also in quality) in order to accurately resolve regions of stratification, and provide quality data for CFD validation.

Ref. 1: E. F. Hicken, K. Verfondern (eds.) "Investigation of the Effectiveness of Innovative Passive Safety Systems for Boiling Water Reactors, Vol. 11, Energy Technology series of the Research Center Jülich, May 2000.

EUBORA

The EU Concerted Action on Boron Dilution Experiments (EUBORA) had 15 partners, with Fortum, Finland as the coordinator. Most of the partners from the FLOMIX-R project (see below) participated also in EUBORA. The project started in late 1998, and finished within about 15 months.

The primary objective was to discuss and evaluate the needs for a common European experimental and analytical programme to validate the calculation methods for assessing transport and mixing of diluted and boron-free slugs in the primary circuit during relevant reactor transients. The second objective was to discuss how the inhomogeneous boron dilution issues should be addressed within the EU.

The partners concluded that there was a clear need to understand the role of mixing in mitigating the consequences of inhomogeneous boron dilution. In particular, the mixing of a boron-reduced slug on its way from the location of formation to the reactor core inlet is important. In order to take full benefit of this mechanism, one should be able to predict the degree of mixing for the reactor case in the most reliable way. Though 3-D CFD methods do provide an effective tool for mixing calculations, it is important to study the slug transportation in sufficient detail, and to perform the calculations under transient conditions. The code calculations, and the applied turbulent mixing models, have to be validated by experiments. Although a number of small-scale and large-scale tests have been performed in existing facilities, the current status of assessment is deemed to be incomplete. In particular, the large-scale experimental database does not cover all the slug motion and mixing cases.

It was also proposed that cooperation among the existing 1/5-scale experiments would provide useful information by focussing on several phenomenological aspects not yet fully covered by the experimental programmes. It was also concluded that other fluid mixing and flow distribution phenomena should be regarded in the same context, since the final aim is to justify and assess the application of CFD codes for general reactor calculations.

Large-scale experiments (scale 1/2) would provide confirmatory data for the existing 1/5-scale experiments, and the partners supported the proposal to modify the existing PANDA facility at PSI for large-scale mixing experiments, though this has yet to be carried out.

Ref. 1. Tuomisto H., *Final Report: EUBORA Concerted Action on Boron Dilution Experiments*, EU Framework Programme on Nuclear Fission Safety, AMM-EUBORA(99)-P002, Dec. 1999.

FLOWMIX-R

Fluid mixing and flow distribution in the reactor circuit (FLOWMIX-R) is an EU 5th Framework shared cost action programme with 11 participants, with the Forschnungszentrum Rossendorf, Dresden responsible for project coordination.

1. Forschungszentrum Rossendorf, Dresden (DE)
2. Vattenfall Utveckling AB, Älvkarleby (SE)
3. Serco Assurance, Dorchester, Dorset (GB)
4. GRS, Garching (DE)
5. Fortum Nuclear Services, Vantaa (Fin)
6. PSL, Villingen (SL)
7. VUJE, Trnava (SK)
8. NRI, Rez (CZ)
9. AEKI, Budapest (HU)
10. NPP Paks, Paks (HU)
11. EDO Gidropress, Podolsk (RU)

The project started in October 2001. The first objective of the project is to obtain complementary data on slug mixing, and to understand in sufficient detail how the slug mixes before it enters the reactor core. (Slug mixing is the most mitigative mechanism against serious reactivity accidents in local boron dilution transients.) The second objective is to utilise data from steady-state mixing experiments and plant commissioning test data, to determine the primary circuit flow distribution, and the effect of thermal mixing phenomena in the context of the improvement of normal operation conditions and structural integrity assessment. The third objective is to use the experimental data to contribute to the validation of

CFD codes for the analysis of turbulent mixing problems. Benchmark calculations for selected experiments are used to justify the application of turbulent mixing models, to reduce the influence of numerical diffusion, and to decrease grid, time step and user effects in CFD analyses.

Due to the large interest of research organisations and utilities from newly associated states (NASs), a NAS extension of the project, incorporating the research institutions VUJE Trnava, NRI Rez (Czech Republic), AEKI Budapest (Hungary) and the nuclear power plant NPP Paks (Hungary), as well as the research and design organisation EDO Gidropress (Russia), as an external expert organisation, has been undertaken.

The work on the project is performed within five work packages.

In WP 1, the key mixing and flow distribution phenomena relevant for both safety analysis, particularly in steam-line-break and boron-dilution scenarios, and for economical operation and structural integrity, have been identified. Based on this analysis, test matrices for the experiments have been defined, and guidelines have been provided for the documentation of the measurement data, and for performing validation calculations with CFD codes.

In WP 2 on slug mixing tests, experiments on slug mixing at the ROCOM and Vattenfall test facilities have been performed, and the measurement data have been made available to the project partners for CFD code validation purposes. Additional slug-mixing tests at the VVER-1000 facility of EDO Gidropress are also being made available. Two experiments on density-driven mixing (one from ROCOM, one from the Fortum PTS facility) have been selected for benchmarking.

In WP 3 on flow distribution in the cold legs and pressure vessel of the primary circuit, commissioning test measurements performed at the Paks VVER-440 NPP have been used for the estimation of thermal mixing of cooling loop flows in the downcomer and lower plenum of the pressure vessel. A series of quasi-steady-state mixing experiments has been performed at the ROCOM test facility. CFD methods are used for the simulation of the flow field in the primary circuit of an operating full-scale reactor, and computed results compared against available measurement data. Conclusions are being drawn concerning the usability and modelling requirements of CFD methods for these kinds of application.

Concerning WP 4 on validation of CFD codes, the strategy of code validation based on the BPGs, and a matrix of CFD code-validation calculations, has been elaborated. CFD validation calculations on selected benchmark tests are being performed. The CFD validation work is shared among the partners systematically on the basis of a CFD validation matrix.

In WP 5, conclusions on flow distribution and turbulent mixing in NPPs will be drawn, and recommendations on CFD applications will be given.

Quality assurance practice for CFD is being applied, based on the ERCOFTAC BPGs, as specified in the ECORA project for reactor safety analysis applications. Serco Assurance and Vattenfall experts are active in the ERCOFTAC organisation. Most of the FLOMIX-R partners are participating also in ECORA, aimed at an assessment of CFD methods for reactor safety analyses. FLOMIX-R is contributing to the extension of the experimental database on mixing, and the application of CFD methods to mixing problems. Recommendations on the use of CFD codes for turbulent mixing problems defined within FLOMIX-R will be fed back to the ECORA and ERCOFTAC BPGs.

First conclusions from the project are that a new quality of research in flow distribution and turbulent mixing inside the RPV has been achieved in the FLOMIX-R project. Experimental data on slug mixing, with enhanced resolution is space and time, has been gained from various test facilities, and covers different geometrical and flow conditions. The basic understanding of momentum-controlled mixing in highly turbulent flows, and buoyancy-driven mixing in the case of density differences between the mixing fluids, has been improved significantly. A higher level of quality assurance in CFD code validation has been achieved by consistently applying BPGs to the solution procedure.

The web address for FLOWMIX-R is http://www.fzd.de/FWS/FLOMIX/

Ref. 1: F.-P. Weiss et al., "Fluid Mixing and Flow Distribution in the Reactor Circuit (FLOWMIX-R)", Proc. FISA-2003/EU Research in Reactor Safety, 10-13 Nov. 2003, Luxembourg.

ASCHLIM

In the Accelerator Driven System (ADS) concept, thermal neutrons produced by bombarding a high-density target with a proton beam, are utilised to produce energy and for the transmutation of radioactive waste. In some designs, the target material is a Heavy-Liquid-Metal (HLM), which also serves as the primary coolant, taking away the heat associated with the spallation reactions that produce the neutrons. Power densities can easily reach 1000 W/cm^3, not only in the liquid metal, but also in critical structures surrounding the spallation region. Structural materials work at very high temperatures, and have to themselves dissipate large quantities of heat. It is essential to have CFD tools capable of reliably simulating the critical phenomena that occur, since it is not possible to experimentally simulate the acquired power densities without actually using a beam.

The ASCHLIM project (Assessment of Computational Fluid Dynamics Codes for Heavy Liquid Metals) is an Accompanying Measure of the Euratom 5th Framework Programme), and aims at joining different experiences in the field of HLMs, both, in the experimental and numerical fields, and creating an international collaboration to (1) make an assessment of the main technological problems in the fields of turbulence, free surface and bubbly flow, and (2) coordinate future research activities.

Where possible, the assessments have been made on the basis of existing experiments, whose basic physical phenomena are analysed through the execution of calculational benchmarks. Selected commercial codes are used, because of their widespread availability, robustness and flexibility. In some particular cases, research codes belonging to particular research institutes have also been considered, given the fact that they often contain state-of-the-art numerical schemes and models. Particular attention is paid in the project to problems associated with turbulence modelling for HLMs, especially those associated with turbulent heat transfer (i.e. uncertainties in specifying the turbulent Prandtl number), free-surface modelling (in the windowless ADS concept, the beam impinges on the liquid surface) and bubbly flows (one ADS design incorporates gas injection to enhance natural circulation).

Some important indications about the use of CFD turbulence models have come from the ASCHLIM benchmarking activity, although in some cases only partial conclusions could be drawn, principally due to the lack of experimental measurements of turbulence quantities. The most important point to be clarified is the exact range of applicability of the turbulent Prandtl number approach to HLM flows, and possibly to extend it through the formulation, if it exists, of a relationship between it and the local fluid and flow characteristics (e.g. molecular Prandtl number and turbulent Reynolds number), valid at least in the range of Peclet numbers of interest for ADS applications.

Further benchmarking exercises in relation to free-surface configurations, and in particular new experiments with water, are recommended. (The use of water as stimulant fluid arises because the measurement possibilities with water are much broader, and less expensive, than with HLMs.) However, the final assessment clearly must involve experiments with the real or very similar fluids (PbBi, Hg).

The need for full 3-D simulations was stressed by most of the participants. However, it must be pointed out here that such simulations could lead to very large, if not prohibitively excessive, CPU times, at least with the present generation of computers. New developments with research codes might also improve the basic knowledge and understanding of free-surface behaviour.

Ref. 1: B. Arien (Ed.) "Assessment of Computational Fluid Dynamics codes for Heavy Liquid Metals", Final Technical Report, October 2003.

EXTRA MATERIAL

Aix-en-Provence, May 2002 Exploratory Meeting

The meeting was in two parts: first, several presentations were given describing CFD applications to relevant NRS issues, and then a working group, under the joint chairmanship of J. C. Micaelli (IRSN) and J. Mahaffy (PSU), was convened, with the purpose of defining an action plan on the "application of CFD to nuclear reactor safety problems". This initiative was followed up at the subsequent IAEA/NEA Technical Meeting in Pisa (see below), where further discussions took place, and became the starting point of the present activity.

The technical presentations covered the areas listed here.

- Recent IRSN work on the application of CFD to primary-system-related phenomena (induced breaks, hot-leg temperature heterogeneity and PTS) and containment-related (development and use of the TONUS code) phenomena.
- The ECORA (Evaluation of Computational Methods for Reactor Safety Analysis) 5^{th} Framework Programme.
- The application of in-house codes at NUPEC to provide the Japanese Regulatory Authority with an independent means of assessment of safety analysis of APWR internals. The issues addressed included flow distribution into the neutron reflector (an innovative design improvement), turbulent flow in the downcomer, γ-heating of the neutron reflector, and flow-induced vibrations.
- Mixing of containment gases (relating to ECORA, ISP-42 activities), aerosol deposition (PHEBEN-2 project), wall condensation, liquid-gas interface tracking, and bubble dynamics in suppression pools.
- Application of CFD techniques associated with various EU projects, including PHEBEN-2, TEMPEST, ECORA and NACUSP.
- The need for two-phase CFD in NRS, including details and preliminary conclusions from the EUROFASTNET project, and the latest R&D developments embodied within the joint CEA/EDF code NEPTUNE.
- Some NRS applications requiring CFD: boron dilution, thermal fatigue, induced pipe rupture, PTS, long-term waste storage, together with latest developments of the CEA code TRIO-U.

All the items covered at this meeting have been identified as topics relevant to the activities of this group, and information concerning them is itemised elsewhere in this report. Consequently, no further explanation is given here. A CD-ROM was prepared of the presentations, but no written papers were required.

Ref. 1: "Exploratory Meeting of Experts to Define an Action Plan on the Application of Computational Fluid Dynamics (CFD) Codes to Nuclear Reactor Safety Problems, Working Group on the Analysis and Management of Accidents", Aix-en-Provence, France, 15-16 May, 2002, NEA/SEN/SIN/AMA(2002)16.

IAEA/NEA Technical Meeting, Pisa, November 2002

The meeting was convened to provide an international forum for the presentation and discussion of selected topics related to various applications of CFD to NRS problems, with the intention to use the material presented to identify further needs for investigation. There were 31 oral and 16 poster session presentations, the principal areas covered being PTS, boron dilution, in-vessel mixing, in-vessel severe accidents, containment studies, combustion and two-phase modelling. Presentations and papers are available on CD-ROM.

Ref. 1: "Technical Meeting on the Use of Computational Fluid Dynamics (CFD) Codes for Safety Analysis of Reactor Systems, including Containment", IAEA-OECD/NEA Joint Meeting, Pisa, Italy, 11-14 November, 2002.

OECD/CSNI Workshop in Barcelona 2000

This was the follow-up meeting to that held at Annapolis in 1996, and was intended to review the developments in the areas which had been identified at that time for special focus, to analyse the present status of current thermal-hydraulic and neutronics codes, and to evaluate the role of such tools in the evolving regulatory environment. Though the focus of the meeting, as at Anaheim, remained on system codes, some time was spent on the emerging role of CFD in NRS issues. In the findings and recommendations, it was recognised that CFD involvement was required in areas where the details of local flow behaviour was of importance, and identified thermal stratification and boron dilution as two such areas.

It was recognised (GRS) that though CFD had its roots outside of the nuclear industry, it was attractive to apply a product with proven capability and a large user community in reactor applications. Of particular advantage is the fact that CFD can be readily applied in regions of geometric complexity, and have the capability of modelling turbulence in those situations where it is the dominant flow mechanism, such as for PTS or containment mixing. Everywhere it was emphasised that the major achievements of CFD are for single-phase flows, and that considerable research effort needs to be expended on the physical modelling side if this success is going to be extended to the two-phase flow situations relevant to NRS problems. Some early advances are cited for dispersed flow and the simulation of nucleate boiling using mechanistic models, and a "concerted action" within Germany was announced, involving research centres, university institutes, GRS, a major code vendor and parts of industry, whereby the code ANSYS-CFX-5 would be further developed for the specific needs of the nuclear industry.

Also emphasised at the Workshop was the need to couple CFD modules with system codes, since it was hardly feasible to model all reactor components using a CFD-type discretisation. Generally, it was recognised that for some important transients (boron dilution and PTS) system codes introduced excessive numerical diffusion, due to the use of first-order difference schemes and coarse meshes, that front-tracking methods in these codes did not improve matters, and that CFD was needed to obtain reliable estimates of the degree of flow mixing taking place.

Otherwise, the capabilities of CFD, and its proven worth in non-nuclear applications, was acknowledged, but that considerably more work on two-phase modelling – meaning closure laws and turbulence – was needed.

Ref. 1: Advanced Thermal-Hydraulic and Neutronic Codes: Current and Future Applications, OECD/CSNI Workshop, Barcelona, Spain, 10-13 April, 2000, NEA/CSNI/R(2001)

5. ESTABLISHED ASSESSMENT BASES FOR NRS APPLICATIONS

5.1 Boron Dilution

Introduction

During boron-dilution events, a volume (slug) of boron-deficient water enters the reactor core after start-up of the main circulation pump, or after recovery of natural circulation. In contrast to the PTS events (see 5.2), the slug fills all the cold leg cross section, and flow rates are usually higher. Experiments generally try to reproduce the mixing in the reactor downcomer and lower plenum, upstream of the reactor core inlets. The main experimental facilities are ROCOM (FZD Rossendorf, Germany), modelling the Konvoi reactor, OKB Gidropress (Russia), modelling the VVER-1000 reactor, and Vattenfall (Sweden), modelling the Westinghouse three-loop reactor. Very detailed results are also available from a series of tests carried out on the University of Maryland four-leg loop, which formed the basis of the OECD/NEA International Standard Problem ISP-43. All these works are referenced at the end of the section, which also cites associated CFD simulations.

University of Maryland experiments and corresponding simulations (ISP-43)

Under the terms of ISP-43, two sets of experiments performed on the University of Maryland facility UM2x4 Loop were made available for numerical analysis. Originally, these for "blind" analyses, but several post-test simulations have been published since then.

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. Quite detailed boundary conditions were provided for the analysts, and time histories of temperatures at nearly 300 positions at eleven levels within the downcomer and lower plenum were available. The problem with wall heat flux was resolved by application of an isolating paint on the wall inner surfaces. The model of the RPV with positions of thermocouples marked is shown in Fig. 5-1.

In Fig. 5-2, a transparent replica of the metallic vessel, the Boron-Mixing Optical Vessel (B-MOV), is also shown. This was used for velocity measurements and flow visualisations utilising Laser Induced Fluorescence (LIF) techniques. Both "front injection" and "slug injection" classes of tests were conducted. From the visualisation, the time development of flow patterns in both cases can be seen.

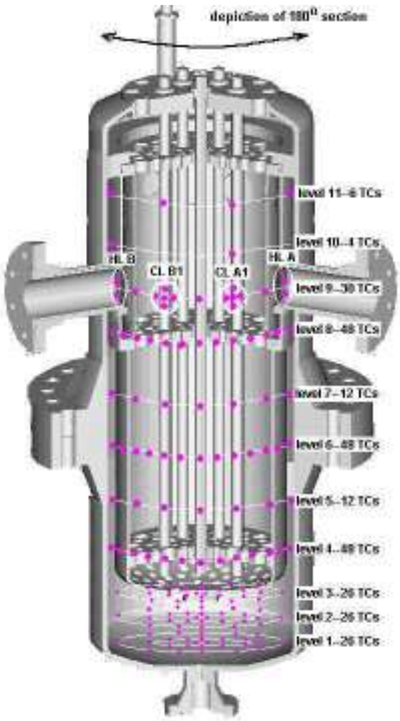


Fig. 5-1: UM 2x4 Loop RPV (integral vessel) and positions of thermocouples.

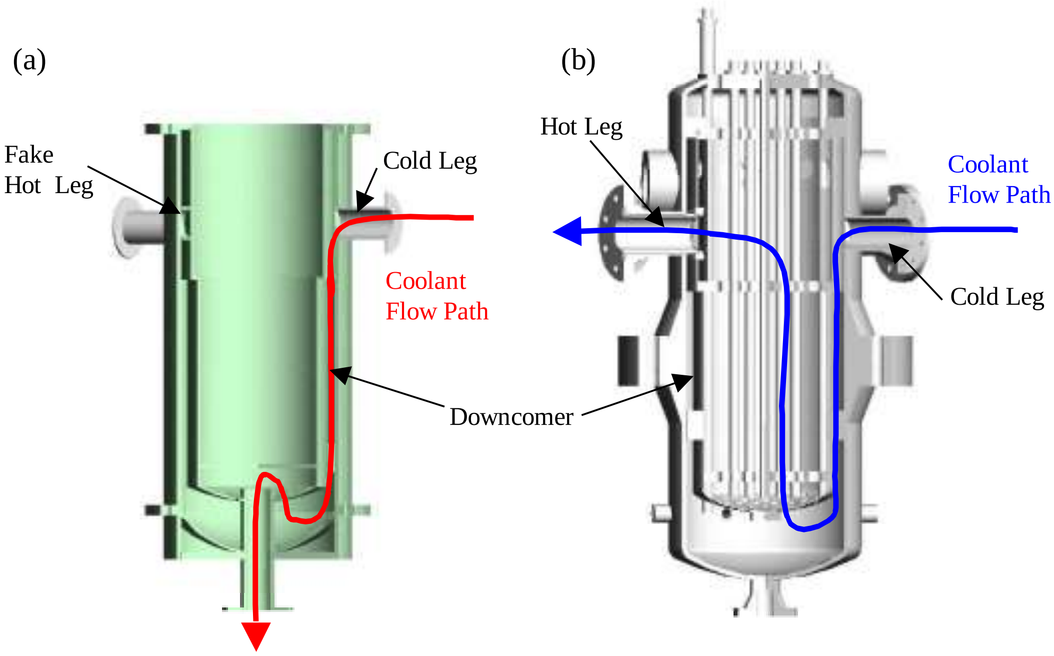


Figure 5-2: (a) B-MOV and (b) integral vessels

One aspect of the results analysed is the possible dependency of the flow pattern in the downcomer on buoyancy. For Fr<6, the incoming flow penetrates downwards in a single jet, whereas for Fr>10 the flow splits into two jets, forming a stagnation region under the point of injection. The two flow patterns were even found for repeated "identical" runs in the critical Froude number range 6<Fr<10. The tests provided very interesting results from visualisation of the flow, which can help in deciding the importance of buoyancy in a given case.

Ten participants from eight countries participated in the blind-calculation phase of the benchmark. The CFD codes featured were ANSYS-CFX-4, ANSYS-CFX-TASCflow, FLUENT and TRIO-U. The time history of the average temperature at the downcomer outlet was selected as the *target variable* for code comparison. Major factors influencing results from the simulations include: choice made for the solution domain (e.g., whether or not to include the core region), position of the outlet and selection of the outlet boundary condition, buoyancy effects, temperature dependency of water properties, modelling of the perforated bottom and core support plate, the distribution, size and type of mesh cells used, inlet boundary condition (uniform velocity, velocity profile, turbulent intensity), turbulence model adopted, order of discretisation schemes for the numerics, time step size, limits of convergence, etc. Comparison of the results of computations with the measured data revealed considerable discrepancy, even among the users of the same code. Some post-test analyses were also carried out, focusing on selected modelling issues such as characteristics of porous-body modelling of the core barrel bottom and core support plates, importance of buoyancy, mesh dependency, etc. It is to be hoped that such analyses will continue, and the results will be made available to the public.

Ref. 1: Gavrilas, M., Hoehne, T.: OECD/CSNI ISP Nr. 43 Rapid Boron Dilution transient tests for code verification post-test calculation with ANSYS-CFX-4. Wissenschaftlich-Technische Berichte. Forschungszentrum Rossendorf FZR-325, Juli 2001.

Ref. 2: Gavrilas, M., Kiger, K.: OECD/CSNI ISP Nr. 43 Rapid Boron-Dilution Transient Tests for Code Verification, September 2000.

Ref. 3: Gavrilas M., Scheuerer M., Tietsch W.: Boron mixing experiments at the 2x4 UMCP test facility. Wechselwirkungen Neutronenphysik und Thermofluiddynamik. Fachtagung der KTG-Fachgruppen “Thermo- und Fluiddynamik” und “Reaktorphysik und Berechnungsmethoden”. Forschungszentrum Rossendorf, January 31 to February 1, 2000, Germany.

Ref. 4: Gavrilas, M., Kiger, K.: ISP-43: Rapid Boron Dilution Transient Experiment. Comparison Report. NEA/CSNI/R(2000)22, February 2001.

Ref. 5: Gavrilas, M., Woods, B. G.: Fr number effects on downcomer flowpattern development in cold leg injection scenarios. Proc. of ICONE10, Arlington 2002, ICONE10-22728.

ROCOM experiments (FLOMIX-R)

In 1998, the Rossendorf test facility ROCOM was constructed for the investigation of coolant mixing phenomena in primary circuits of PWRs. ROCOM is a 1:5 scaled Plexiglas model of the German PWR Konvoi, consisting of four loops, and with fully controllable coolant pumps. The facility is operated with demineralised water at normal conditions. The coolant mixing is investigated by the injection of slugs of a tracer solution (diluted salt) into the main flow of one loop. The salt concentration is measured by means of wire mesh conductivity sensors with high resolution in time and space. Sensors are installed in the cold leg inlet nozzle of the disturbed loop (256 measuring points), two in the downcomer, just below the inlet nozzles and before the entrance into the lower plenum (2×256 measuring points). The fourth sensor is integrated into the lower core support plate and has one measuring position at each fuel element position. Further, all four outlet nozzles was equipped with sensors (4×256 measuring points). LDA was applied for velocity measurements. The tracer concentration fields established by coolant mixing under stationary and transient flow conditions were then investigated. A general view of the facility is in Fig. 5-3 and the Plexiglas model is shown in Fig. 5-4.

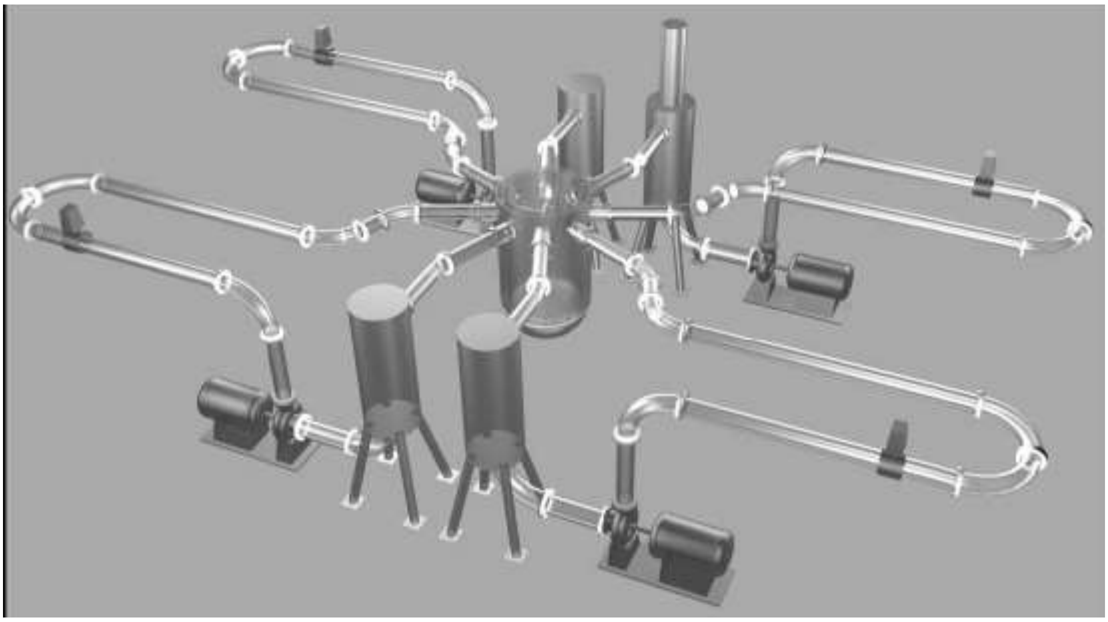


Fig. 5-3: General view of the ROCOM facility.

Fig. 5-4: ROCOM Plexiglas model.

Four different groups of mixing scenarios were investigated:

1. Flow distribution measurements at constant flow rates in the primary circuit. The mass flow rate, the number of operating loops, the status of non-operating loops (reverse flow or closed) and the friction losses at the core inlet were all varied. These scenarios cover steam line break accidents. Averaged data for a quasi-stationary state were used, to gain mixing coefficients at the core inlet. The experiments showed that, even for the turbulent flow in the reactor vessel (downcomer, lower plenum, core, upper plenum), the mixing of a disturbance in one loop remains incomplete for all the cases investigated. For the case of four-loop operation, the influence of perturbations of temperature or boron concentrations in one loop is mainly concentrated in the corresponding 90° sector of the

core inlet. Maximum mixing coefficients of about 90% were obtained in that case.

2. Slug mixing experiments with a change of the flow rate in one or several loops. This event might happen during boron dilution transients by an inadvertent start of a main coolant pump, with coolant having reduced boron concentration, or by start of natural circulation following refilling after a small break LOCA. After start of a main coolant pump, the deborated coolant in the loop first appears at the core inlet on the opposite side to injection. During the transient, the perturbation at the core inlet moves gradually to the side of the disturbed loop. This behaviour is caused by secondary turbulent vortices in the downcomer, whose structure has been measured using LDA.

3. Density-driven experiments, which correspond to scenarios with the injection of cold Emergency Core Cooling (ECC) water (increased density) into the cold leg, and incomplete mixing on the way to the core. The flow of coolant in the downcomer may lead to pre-stressed thermal shock events. The critical values of the Froude Number for the transition from momentum-driven to density-driven flow were determined. Mixing experiments with reduced density were also performed.

4. Mixing experiments for determining of the relation between temperature and boron dilution distribution at the reactor outlet, i.e., the upper plenum, were also performed. For these, the coolant from one certain fuel element to the sensors in the four outlet nozzles was measured. Experiments for all fuel elements of a 90° symmetry sector of the core were performed and stationary mixing coefficients at each of the 864 measuring points were determined. By means of these coefficients, the temperature or boron dilution profile in the outlet nozzles can be reconstructed.

Matrix of slug mixing tests performed at the ROCOM test facility is in the following Table.

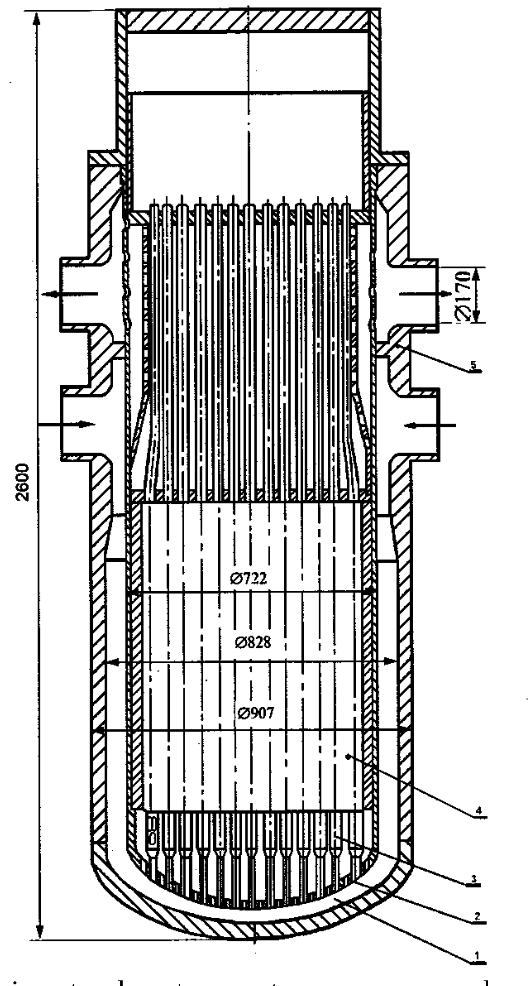
Run	Ramp length (s)	Final volume flow rate (m^3/h)	Slug volume (m^3)*	Initial slug position (m)*	Status of unaffected loops
ROCOM-01	14	185.0	40.0	10.0	Open
ROCOM-02	14	185.0	20.0	10.0	Open
ROCOM-03	14	185.0	4.0	10.0	Open
ROCOM-04	14	185.0	4.0	2.5	Open
ROCOM-05	14	185.0	4.0	22.5	Open
ROCOM-06	14	185.0	4.0	40.0	Open
ROCOM-07	14	185.0	20.0	10.0	Closed
ROCOM-08	28	92.5	4.0	10.0	Open
ROCOM-09	56	46.3	4.0	10.0	Open
ROCOM-10	14	148.0	4.0	10.0	Open
ROCOM-11	14	222.0	4.0	10.0	Open
ROCOM-12	14	185.0	8.0	10.0	Open

* related to the original reactor

A comprehensive knowledge base on mixing phenomena in nuclear power reactors and an experimental database has been created around these experiments, which is well suited for CFD code validation. Simulations been carried out using the codes ANSYS-CFX-4, ANSYS-CFX-5 and TRIO_U using a variety of turbulence modelling options. The ANSYS-CFX-5 simulation used the RSM turbulence model, whereas the TRIO_U simulation used an LES approach. It was concluded that both simulations required approximately the same CPU time since ANSYS-CFX-5 used large time steps (implicit scheme), but RSM requires the solution of many transport equations. The LES approach uses smaller time steps, but a smaller number of equations is solved. The results of LES seem to be slightly better at both the upper and lower downcomer planes. DES (Detached Eddy Simulation) approach will be tested in the next step.

Gidropress Facility (FLOMIX-R)

Three tests were performed on the OKB Gidropress experimental facility (Fig. 5-5) with different final flow rates: 225 m3/h (6 runs), 640 m3/h (8 runs), and 800 m3/h (6 runs). Temperatures at the reactor core inlet were measured and the results were provided to the FLOMIX-R participants. Selected tests were then simulated within the FLOMIX-R project with the ANSYS-CFX-5 and FLUENT computer codes. Some problems with uncertainty of the measured quantities (loop flow rates) and with probable, but unknown, wall heat transfer caused differences between measured data and numerical predictions. Improved results were obtained once the walls were explicitly modelled, but solution of conjugate heat transfer problems is much more demanding in terms of computer memory and CPU time. This is probably



a common problem of all experiments where temperatures are measured.

Fig. 5-5: Gidropress facility – model of the reactor

Vattenfall Experiments (FLOMIX-R)

The Vattenfall experiments are similar to the OKB Gidropress tests; in both cases, a slug of finite volume enters the reactor core. Measurements of concentrations at the “core” inlet and velocities in the downcomer for four transient cases, VATT-01 (large slug), VATT-02 (medium-sized slug), VATT-03 (small slug) and VATT-04 (slow transient), were planned within the FLOMIX-R project.

Both steady-state (only velocity field calculated) and transient simulations were made for VATT-02 within the project by several groups using the FLUENT and ANSYS-CFX-5 codes. A schematic of the facility is given in Fig. 5-5.

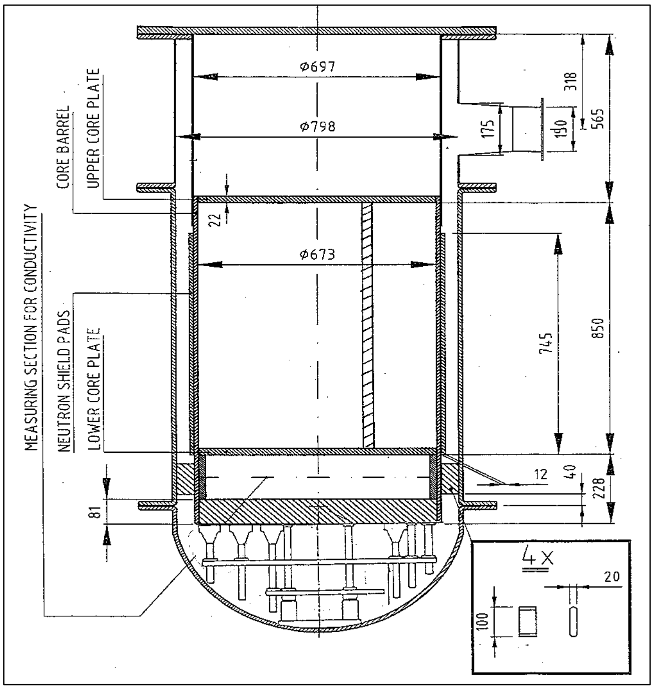


Fig. 5-5: Vattenfall test facility: reactor vessel

A matrix of the slug mixing tests is given in the following Table.

Run	Ramp length (s)	Final volume flow rate (m^3/h)	Slug volume (m^3)*	Initial slug position (m)*	Status of unaffected loops
VATT-01	16	429	14.0	10.0	Open
VATT-02	16	429	8.0	10.0	Open
VATT-03	16	429	4.5	10.0	Open
VATT-04	40	172.8	8.0	10.0	Open

* related to the original reactor

Thorough review of the boron dilution experiments has been undertaken. Reynolds number scaling effects have been investigated, showing that the effects are quite small for the flow rates used in the tests. It was concluded from the tests that the structures in lower plenum have a significant influence on the mixing of the slug. Analysis of the tests for which concentration measurement, velocity measurement and visualization for two different slug sizes and several Reynolds numbers were obtained was carried out within the FLOMIX-R project.

Ref. 1: Alavyoon, K.: Numerical approach to rapid boron dilution transients for a PWR mock-up – I. Grid dependence studies of the flow field. US 95:34, Vattenfall, 1995.

Ref. 2: Alavyoon, F., Hemstroem, B., Andersson, N.-G., Karlsson, R. I.: Experimental and computational approach to investigating rapid boron dilution transients in PWRs. OECD Specialist Meeting on Boron Dilution Reactivity Transients, State College, PA, USA, October 18-20, 1995.

Ref. 3: Almenas, K. K., Dahlgren, C. N., Gavelli, F., DiMarzo, M.: Numerical diffusion issues in the evaluation of boron mixing using the COMMIX code. MD-NUME-98-02.

Ref. 4: Alvarez, D. et al.: Three-dimensional calculations and experimental investigations of the primary coolant flow in a 900 MW PWR vessel. NURETH-5, Salt Lake City, Sept. 1992, Vol. II, pp. 586 – 592.

Ref. 5: Andersson N.G., Hemström B., Karlsson R.I. & Jacobson S. "Physical modelling of a Rapid Boron Dilution Transient." Proceedings of Nureth 7, Saratoga Springs, USA, 1995.

Ref. 6: Bezrukov, Yu. A., Logvinov, S. A.: Some experimental results related to the fast boron dilution in the VVER-1000 scaled model. Presented in the 3rd Workshop Meeting of the EUBORA project, PSI, Switzerland, 1999 (internal EUBORA document).

Ref. 7: Bezrukov, Yu. A.: Documentation on slug mixing experiments of OKB Gidropress. Presented at 3rd FLOMIX-R Meeting, PSI, 2002 (FLOMIX-R internal document).

Ref. 8: Bieder U., Fauchet G., Bétin S., Kolev N., Popov D.: Simulation of mixing effects in a VVER-1000 reactor. The 11th Int. Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11), Avignon, France, October 2-6, 2005. Paper 201.

Ref. 9: Boros, I., Aszodi, A.: Numerical analysis of coolant mixing in the RPV of VVER-440 type reactors with the code ANSYS-CFX-5.5.1. Technical Meeting on Use of Computational Fluid Dynamics (CFD) Codes for Safety Analysis of Reactor Systems, Including Containment. Pisa, Italy, 11-14 November 2002.

Ref. 10: C.R. Choi, T.S. Kwon and C.H. Song, "Numerical analysis and visualization experiment on behavior of borated water during MSLB at the RCP running mode in an advanced reactor", Nuclear Engineering and Design, to appear.

Ref. 11: Dury, T.: CFD simulation of steady-state conditions in a 1/5th-scale model of a typical 3-loop PWR in the context of boron dilution events. Technical Meeting on Use of Computational Fluid Dynamics (CFD) Codes for Safety Analysis of Reactor Systems, Including Containment. Pisa, Italy, 11-14 November 2002.

Ref 12: Elter, J.: Experimental investigation of thermal mixing phenomena in a six-loop VVER type reactor vessel. Summary report.

Ref. 13: Gango, P.: Application of numerical modelling for studying boron mixing in Loviisa NPP. OECD/CNSI Spec. Meeting on Boron Dilution Reactivity transients. State College PA USA, Oct. 18-20, 1995.

Ref. 14: Gango, P.: Numerical boron mixing studies for Loviisa nuclear power plant. Nucl. Eng. Design 177(1997), 239-254.

Ref. 15: Gavrilas, M., Hoehne, T.: OECD/CSNI ISP Nr. 43 Rapid Boron Dilution transient tests for code verification post-test calculation with ANSYS-CFX-4. Wissenschaftlich-Technische Berichte. Forschungszentrum Rossendorf FZR-325, Juli 2001.

Ref. 16: Gavrilas, M., Kiger, K.: OECD/CSNI ISP Nr. 43 Rapid Boron-Dilution Transient Tests for Code Verification, September 2000.

Ref. 17: Gavrilas M., Scheuerer M., Tietsch W.: Boron mixing experiments at the 2x4 UMCP test facility. Wechselwirkungen Neutronenphysik und Thermofluiddynamik. Fachtagung der KTG-Fachgruppen "Thermo- und Fluiddynamik" und "Reaktorphysik und Berechnungsmethoden". Forschungszentrum Rossendorf, January 31 to February 1, 2000, Germany.

Ref. 18: Gavrilas, M., Kiger, K.: ISP-43: Rapid Boron Dilution Transient Experiment. Comparison Report. NEA/CSNI/R(2000)22, February 2001.

Ref. 19: Gavrilas, M., Woods, B. G.: Fr number effects on downcomer flowpattern development in cold leg injection scenarios. Proc. of ICONE10, Arlington 2002, ICONE10-22728.

Ref. 20: Grunwald, G., Hoehne, T., Kliem, S., Prasser, H.-M., Rohde, U.: Status report on R&D activities on boron dilution problems. Presented at the 1st EUBORA project Meeting, Vantaa, Finland, 1998 (EUBORA internal document).

Ref. 21: Grunwald, G., Hoehne, T., Prasser, H.-M.: Investigation of coolant mixing in pressurized water reactors at the Rossendorf mixing test facility ROCOM. 8th International Conference on Nuclear Engineering (ICONE8), Baltimore, USA, 2000.

Ref. 22: Grunwald, G.; Höhne, T.; Kliem, S.; Prasser, H.-M.; Rohde, U.; Weiß, F.-P. (2002): Experiments and CFD Calculations on Coolant Mixing in PWR – Application to Boron Dilution Transient Analysis. TECHNICAL MEETING on Use of Computational Fluid Dynamics (CFD) Codes for Safety Analysis of Reactor Systems, including Containment, Pisa, Italy, 11–15 November 2002

Ref. 23: Grunwald, G.; Höhne, T.; Kliem, S.; Prasser, H.-M.; Rohde, U.; Weiß, F.-P.: Coolant mixing studies for the analysis of hypothetical boron dilution transients in a PWR, 11th International Conference on Nuclear Engineering ICONE-11, Tokyo, Japan, April 2003

Ref. 24: Hemstroem B. et al.: Validation of CFD codes based on mixing experiments (Final report on WP4). EU/FP5 FLOMIX-R Report, FLOMIX-R-D11, Vattenfall Utveckling (Sweden), 2005.

Ref. 25: Hemstroem, B., Andersson, N.-G.: Physical modelling of a rapid boron dilution transient. The EDF case. Report VU-S 94:B16, Vattenfall 1994.

Ref. 26: Hemstroem, B., Andersson, N.-G.: Physical modelling of a rapid boron dilution transient. – I. Reynolds number sensitivity study for the Ringhals case. Report US 95:5, Vattenfall 1995.

Ref. 27: Hemstroem, B., Andersson, N.-G.: Physical modelling of a rapid boron dilution transient – II. Study of the Ringhals case, using a more complete model. Report US 97:20, Vattenfall 1997.

Ref. 28: Hoehne T.: Numerical modelling of a transient slug mixing experiment at the ROCOM test facility using ANSYS-CFX-5. The 11th Int. Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11), Avignon, France, October 2-6, 2005. Paper 481.

Ref. 29: Hoehne, T., Grunvald, G., Rohde, U.: Coolant mixing in Pressurized Water Reactors. Proc. of the 8th AER Symposium on VVER Reactor Physics and reactor Safety, Bystrice nad Pernstejnem, Czech Republic, September 21 – 25, 1998.

Ref. 30: Hoehne T., Kliem S., Bieder U.: Modelling of a buoyancy-driven flow experiment at the ROCOM test facility using the CFD codes ANSYS-CFX-5 and Trio_U. Nucl. Eng. Design 236 (2006) 1309-1325.

Ref. 31: Hoehne, T., Rohde, U., Weiss, F.-P.: Experimental and numerical investigation of the coolant mixing during fast deboration transients. 9th AER Symposium on VVER reactor physics and reactor safety, Demanovská Dolina, Slovakia, Oct. 4-8, 1999.

Ref. 32: Kiger, K. T., Gavelli, F.: Boron mixing in complex geometries: flow structure details. Nucl. Engineering and Design 208 (2001), 67 – 85.

Ref. 33: Kim, J. H.: Analysis of Oconee Unit 1 downcomer and lower plenum thermal mixing tests using COMMIX-1A. EPRI NP-3780, November 1984.

Ref. 34: Kliem, S., Hoehne, T., Weiss, F.-P., Rohde, U.: 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 ANSYS-CFX-4. NURETH 9, San Francisco, USA 1999.

Ref. 35: Menant, B: Simulations numériques fines de la thermohydraulique monophasique des circuits primaires de Réacteurs à Eau Pressurisée.Cours INSTN CEA Grenoble « Ecoulements et transferts de chaleur monophasiques » 20-24 mars 2000.

Ref. 36: Prasser, H.- M.; Grunwald, G.; Höhne, T.; Kliem, S.; Rohde, U.; Weiss, F.-P.: Coolant mixing in a PWR - deboration transients, steam line breaks and emergency core cooling injection - experiments and analyses, Nuclear Technology 143 (2003) 37 – 56.

Ref. 37: Rohde, U., Kliem, S. Toppila, T., Hemstroem, B., Cvan, M., Bezrukov, Y., Elter, J., Muehlbauer, P.: Identification of mixing and flow distribution key phenomena. FLOMIX-R Project Deliverable D2 (internal document). 2002.

Ref. 38: Rohde U., Kliem S., Hoehne T., Prasser H.-M., Hemstroem B., Toppila T., Elter J., Bezrukov Y., Scheuerer M.: Measurement data base on fluid mixing and flow distribution in the reactor circuit. The 11th Int. Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11), Avignon, France, October 2-6, 2005. Paper 258.

Ref. 39: Rohde U., Kliem S., Hoehne T., Karlsson B., Hemstroem B., Lillington J., Toppila T., Elter J., Bezrukov Y.: Fluid mixing and flow distribution in the reactor circuit, measurement data base. Nucl. Eng. Design 235 (2005a) 421-443.

Ref. 40: Schaffrath A., Fischer K.-C., Hahn T., Wussow S.: Validation of the CFD code FLUENT by post test calculation of the ROCOM experiment T6655_21. The 11th Int. Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11), Avignon, France, October 2-6, 2005. Paper 141.

Re**f. 41**: Scheuerer, M.: Numerical simulation of OECD/NEA International Standard Problem No. 43 on Boron Mixing in a Pressurized Water Reactor. Report GRS GmbH.

Ref. 42: Scheuerer, M.: Simulation of OECD/NEA International Standard problem No. 43 on boron mixing transients in a pressurized water reactor. Technical Meeting on Use of Computational Fluid Dynamics (CFD) Codes for Safety Analysis of Reactor Systems, Including Containment. Pisa, Italy, 11-14 November 2002.

Ref. 43: Tinoco, H., Hemstroem, B., Andersson, N.-G.: Physical modelling of a rapid boron dilution transient. Report VU-S 93:B21, Vattenfall 1993.

Ref. 44: Toppila, T.: Experiences with validation of CFD methods for pressure vessel downcomer mixing analyses. Technical Meeting on Use of Computational Fluid Dynamics (CFD) Codes for Safety Analysis of Reactor Systems, Including Containment. Pisa, Italy, 11-14 November 2002.Lillington, J. N. (ed.): PHARE Project PH2.08/95 Prevention of Inadvertent Primary Circuit Dilution. Report AEAT – 4026, Issue 2, January 1999.

Ref. 45: Um, K. – S., Ryu, S. – H., Choi, Y. - S., Park, G. – C.: Experimental and computational study of the core inlet temperature pattern under asymmetric loop conditions. Nucl. Technology 125 (1999), 305 – 315.

Ref. 46: Umminger, K., Kastner, W., Liebert, J., Mull, T.: Thermal hydraulics of PWRS with respect to boron dilution phenomena. Experimental results from the test facilities PKL and UPTF. Nucl. Eng. Design 204 (2001) 191 – 203.

5.2 Pressurised Thermal Shock

A review of PTS-relevant experiments and numerical simulations should start with a quote from the document entitled “Guidelines on Pressurized Thermal Shock Analysis for WWER Nuclear Power Plants. Rev. 1”, AEA-EBP-WWER-08, Dec. 2001:

An important feature of some PTS transients is flow stagnation in the primary circuit. In such a case, the flow distribution is governed by buoyancy forces, i.e. thermal stratification and mixing of cold high-pressure injection water to the cold legs become dominant effects. These phenomena are not predicted correctly with the existing thermal hydraulic system codes.

An extensive experimental database exists for thermal fluid mixing that is relevant to PTS issue, Theofanous, Yan (1991). In this document, the following facilities and experimental runs are summarized:

- Creare 1:5 (Tests 100, 101, 103, 104, 106) and 1:2 (tests May 105, May 106), USA
- IVO (FORTUM) 2:5 (Tests T9, T10, T12, T16, T44, T45, T47, T51, T106, T111 to T116), Finland
- Purdue 1:2 (Runs 0-1C, 0-1C-R, 0IV, 0-2C, 0-2C-R, 0-2V, CE-1C, W-1C,B&W-1C, CE-2C, W-1C-90, CE-1C-PS, CE-3C-0), USA
- HDR 1:1 (Tests T32.11 to T32.15, T32.18 to T32.22, T32.31 to T32.34, T32.36, T32.41, T32.51, T32.52, T32.57, T32.58, T32.61), Germany
- UPTF 1:1 (Runs 020, 021, 023, 025, 026), Germany

According to the ECORA Best Practice Guidelines, experimental data for validation of a CFD code should be complete (geometry, boundary and initial conditions, well analysed as to the physical phenomena involved), high quality (accurate within given error bounds, repeatable, consistent) and publicly available. The data in this database are available only in graphical form; and there are only references to reports with the detailed descriptions of geometry and instrumentation. The document is intended for validation of the REMIX/NEWMIX computer codes, so only limited data are present. In the present form, the database does not meet the BPG for validation of a CFD computer code, but could be used for demonstration computations. For validation, the original reports referenced in the Theofanous, Yan (1991) and cited below must be used.

The following reports describe the CREARE 1:5 tests: Rothe, Ackerson (1982), Fanning, Rothe (1983), Rothe, Marscher (1982) and Rothe, Fanning (1982, 1983).

IVO (FORTUM) tests are described in Mustonen (1984), Tuomisto, Mustonen (1986, 1986a), Tuomisto (1986) and Tuomisto (1987).

Tests on the Purdue facility are described in Theofanous et al. (1984), Iyer et al. (1984), Iyer, Theofanous (1991), Theofanous et al. (1986), Theofanous et al. (1984) and Iyer (1985).

For the CREARE 1:2 test, the following reports are available: Dolan, Valenzuela (1985), and Valenzuela, Dolan (1985).

Some HDR tests are described in Wolf et al. (1984, 1986), Wolf, Schygulla (1985) and Tenhumberg, Wenzel (1985). Further information on experimental results from HDR facility is in Theofanous et al. (1992).

Reports on some UPTF tests are: Sarkar, Liebert (1985), Weiss (1986, 1986a) and Weiss et al. (1987, 1987a).

Some characteristics of selected experimental facilities mentioned above are in Table 4.1, taken over from Wolf et al. (1988).

	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
Cold leg diameter (mm)	143	343	363.5	194	190
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

During a PTS, several more local physical processes can be seen. The corresponding physical models must be validated. A list of such phenomena is contained in Scheuerer (2002) and Pigny (2002). The list is reproduced here since the selected suitable validation experiments should cover at least one of the items of the list:

- impingement of single-phase flow jets;
- impingement of two-phase jets;
- impinging jet heat transfer;
- turbulent mixing of momentum and energy in and downstream of the impingement zone;
- stratified two-phase flow (or free surface flow) within ducts;
- phase change at the steam-water interface (condensation, evaporation);
- rapid transients.

According to the verification and validation philosophy adopted within the ECORA project, also carefully selected separate effect tests were admitted for code verification. Then, the following (single-phase) verification tests were selected, Scheuerer (2002):

- gravitational oscillations of water in a U-shaped tube, see Ransom (1992)
- centralized liquid sloshing in a cylindrical pool, see Maschek et al. (1992)
- single-phase water hammer, see Simpson (1989)

As a single-phase validation test, the following experiment was selected:

- axisymmetric single-phase air jet in air environment, impinging on a heated flat plate, see Baughn, Shimizu (1989).

Validation simulations performed within the ECORA project are summarized in the report Egorov (2004), which is available at http://domino.grs.de/ecora/ecora.nsf, Public Docs.

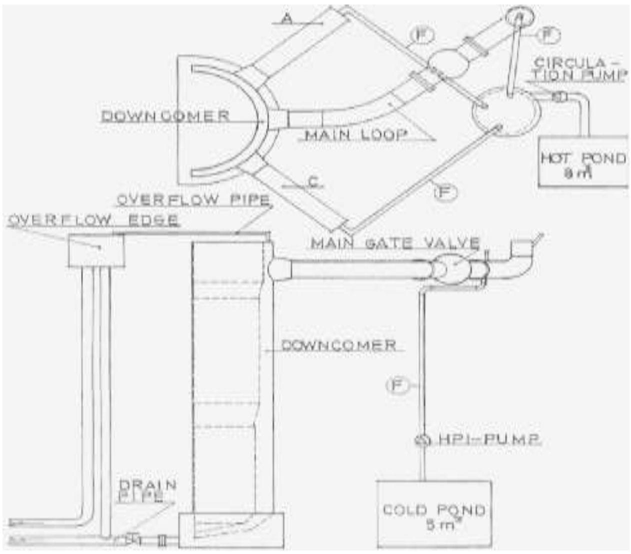
Another region with possible substantial mixing is the sudden change of the reactor downcomer width. This situation is close to the classic CFD benchmark – the backward-facing step. The relevant experimental data can be found in Armaly et al. (1983), and some indications are also in Freitas (1995). For low-Reynolds number situations, DNS data in Lee, Moin (1992) can be also used.

Some further relevant literature on experiments with vertical buoyant plumes or jets is in Kotsovinos (1975) and in Chen, Rodi (1980).

Experimental data on normally impinging jet from a circular nozzle is available in the ERCOFTAC database – Classic Collection. The relevant paper is Cooper et al. (1993).

IVO (FORTUM) test facility

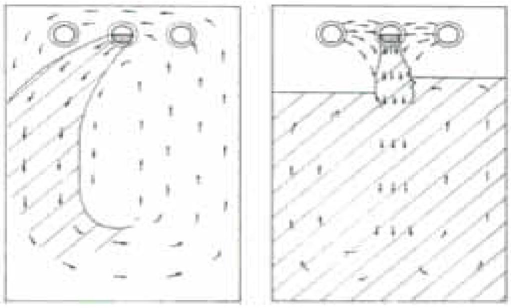
Within the FLOMIX-R project (5th EU Framework Programme), the computer codes FLUENT and ANSYS-CFX were validated against Tests 10, 20 and 21, from the IVO (FORTUM) test facility; see Rohde et al. (2004). A diagram of the FORTUM PTS test facility is shown below. Experimental results from IVO (FORTUM) test facility can also be found in Tuomisto (1987a) from which the Table below showing the test matrix of the thermal mixing programme is reproduced. Later, the facility was reconstructed within the IVO – USNRC PTS information exchange and now has asymmetric orientation of the cold legs and injection nozzles at the top of the cold legs.



test n°	Q_{HPI} l/s	$Q_{L,A}$ l/s	$Q_{L,B}$ l/s	$Q_{L,C}$ l/s	$Fr_{CL,HPI}$	salinity $\Delta\rho/\rho$
3	2.31	0	1.87	0	0.379	0.02
4	2.31	1.87	1.87	1.87	0.376	0.02
7	2.02	1.87	1.87	1.87	0.129	0.16
8	2.00	0	1.87	0	0.129	0.16
9	2.02	0	0	0	0.130	0.16
10	2.31	0	0	0	0.147	0.16
12	0.62	0	0	0	0.040	0.16
13	0.62	0.62	0.62	0.62	0.040	0.16
14	0.62	0	0.62	0	0.039	0.16
15	0.62	1.87	1.87	1.87	0.040	0.16
16	0.31	0	0	0	0.020	0.16
19	0.10	0.3	0	0.3	0.006	0.16
20	2.31	1.87	0	1.87	0.146	0.16
21	2.31	1.87	1.87	1.87	0.147	0.16
22	4.0	2.0	0	2.0	0.253	0.16
23	0.20	0.3	0	1.0	0.013	0.16
26	0.62	1.0	0	1.5	0.096	0.02
27	0.62	0.62	0.62	0.62	0.101	0.02
28	0.20	0.3	0	1.0	0.032	0.02
30	1.25	1.87	0	1.87	0.202	0.02
31	0.62	1.87	0	1.87	0.100	0.02
32	0.10	0.3	0	0.3	0.016	0.02
33	4.0	2.0	0	2.0	0.646	0.02
34	1.25	1.87	0	1.87	0.126	0.06
35	2.31	1.87	0	1.87	0.188	0.10
36	0.62	1.87	0	1.87	0.050	0.02
38	1.25	1.87	0	1.87	0.102	0.02
40	0.20	0.3	0	1.0	0.016	0.02
41	1.25	1.87	0	1.87	0.080	0.13
42	1.25	1.87	0	1.87	0.080	0.16
43	4.0	2.0	0	2.0	0.323	0.02
44	4.0	0	0	0	0.324	0.02
45	4.0	0	0	0	0.255	0.16
46	3.0	2.0	0	2.0	0.477	0.02
47	4.0	0	0	0	0.644	0.02
48	2.0	1.87	0	1.87	0.318	0.02
49	2.31	1.87	0	1.87	0.366	0.02
50	0.62	0	0.62	0	0.100	0.02

The original facility was constructed for study of thermal mixing phenomena in the Loviisa VVER-440 reactor during overcooling transients. It represents a 2:5 scale model of one half of the Loviisa reactor downcomer, with three loops and bottom injection into one loop. The pictures of cold plumes reproduced here are taken from Toppila (2002).

Gango (1995) validated the PHOENICS code against data from these tests. Since the facility is made of transparent material with limited maximum temperature difference, salt was added in some runs to increase the density differences. Three tests were selected for validation: Test 22 and Test 33 differed by $Fr_{CL,HPI}$ and salinity; Test 47 was performed with stagnated loop flow (see Table). Altogether, nine variants of computations were performed, differing in inlet turbulent intensity, order of the discretization of convection terms, time step, and turbulent Prandtl number.

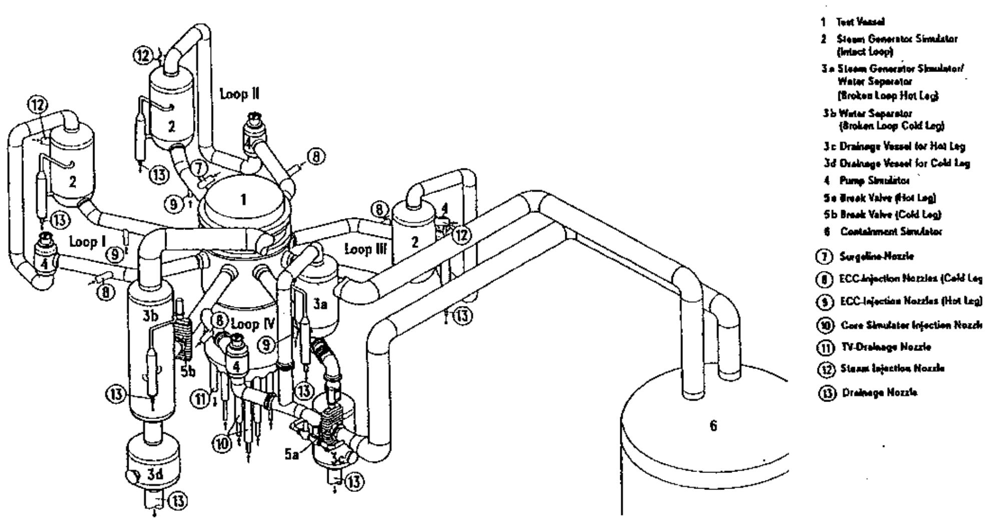


Mixing Test 20 was analysed by Toppila (2002). The model he used had 283000 cells and included also the cold legs with safety injection line. The thermal stratification in the cold leg and reactor downcomer was examined, and the asymmetrical stratification under the cold leg corresponds to the experimental results.

51	2.31	0	0	0	0.372	0.02	
52	0.62	0	1.87	0	0.100	0.02	

UPTF facility

Within the ECORA project, two almost industrial-scale tests were proposed, based on the UPTF experimental facility: UPTF Test 1, and UPTF Test 8 (this test case is available in the OECD/NEA Data Bank http://www.oecd-nea.org/html/dbprog/ccvm/). A schematic of this facility is given here



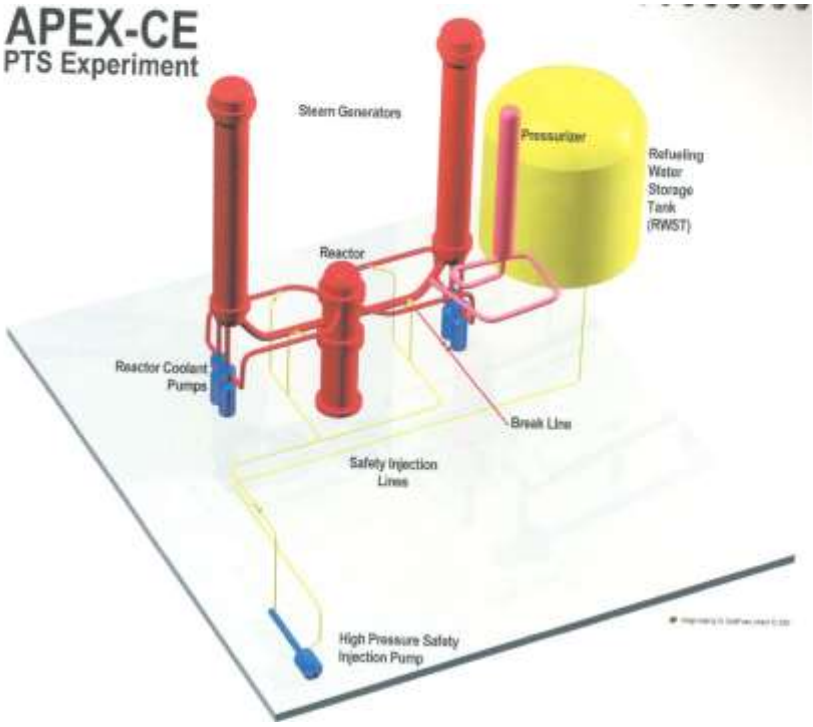
The UPTF Test 1 was simulated by Willemsen, Komen (2005). In this test, the primary system was initially filled with stagnant hot water at 190°C. The cold ECC water, at 27°C, was injected into one cold leg with mass flow rate of 40 kg/s. The authors found that the location of the cold plume along the downcomer thickness depended on modelling of buoyancy as well as on other modelling details. For example, inclusion of detailed models of internals, which should improve the results since it is closer to reality, led to the cold ECC water flowing primarily along the core barrel, whereas an alternating hot and cold fluid was seen to pass the core barrel and vessel wall in the experiment. As a result, the cooling of the RPV wall is significantly underestimated in the computation (by about 50%). These, of course, represent non-conservative results, and should be ignored.

ROCOM test facility

As mentioned in Chapter 3, some experiments with simulated ECC cold water injection were performed in the ROCOM facility. Higher density of water was obtained by addition of glucose, and sodium chloride was used as the tracer. Mass flow was varied between 0 and 15% of the nominal flow rate (the order of magnitude of natural circulation); the density difference was between 0 and 10%. Altogether, 18 experiments were performed, covering density-dominated flows, momentum-dominated flows, and the transition region. A short description of the experiments and numerical simulation of one case with the ANSYS-CFX-5 computer code can be found in Hoehne et al. (2005). Experiments are also described in Rohde et al. (2005), as mentioned in Chapter 3.

APEX Test Facility

The APEX Test Facility at Oregon State University (OSU) was used to perform a series of separate effects and integral systems overcooling tests that examine the conditions that lead to primary loop stagnation and cold leg thermal stratification, see Reyes et al. (2001). The thermal hydraulic phenomena of specific interest are the onset of loop stagnation, the onset of thermal stratification in the cold legs, and characterization of thermal fluid mixing and heat transfer in the downcomer. The former design of the facility was based on the Westinghouse AP600 reactor and a summary of the non-proprietary results is given in Reyes et al. (1999). The present facility APEX-CE simulates the Combustion Engineering Palisades NPP. The modification included the addition of four cold-leg loop seals and HPI nozzles.



The objective of the APEX-CE experimental program was the removal of some conservatism and uncertainties in the earlier PTS study at OSU: like more realistic prediction of the onset of loop stagnation, and effects of asymmetric loop flow. Careful scaling based on PTS phenomena and identification ranking table (PIRT) should ensure that the tests on APEX-CE facility adequately simulate the basic PTS phenomena on the Palisades NPP: natural circulation, primary system depressurisation, secondary system depressurisation, and thermal fluid mixing in the cold legs and downcomer. Both integral system and separate effect tests have been planned. The integral system tests include a series of main steam-line break (MSLB) tests, small hot leg loss-of-coolant accidents (SBLOCAs), and stuck open pressurizer PORV tests. These tests were performed to examine their potential for overcooling the primary side. The conditions for the onset of loop stagnation will be identified and the primary side pressure and temperature time course will be recorded. The separate effect tests will examine the details of cold leg and downcomer fluid mixing under low and stagnant primary loop flow conditions. Fluid temperature profiles in the cold leg and downcomer will be measured as well as the local heat flux and wall temperatures. The data have been analysed using the RELAP5, STAR-CD and REMIX computer codes.

Young, Reyes (2001) compare STAR-CD calculations with APEX-CE test data. Two parametric tests, OSU-CE-0003E and OSU-CE-0003G were selected for the comparison. During the tests, there was natural circulation in the cold leg. The computational model consisted of 839 348 cells and included two cold legs with loop seal, reactor downcomer and lower plenum. The computed results compared well with the APEX-CE data.

One interesting problem connected to the thermal-hydraulic analyses of the pressurized thermal shock is the possibility of interaction of the neighbouring cold plumed in the reactor downcomer. Such interaction was observed in the IVO (FORTUM) tests and was studied also on the APEX-CE facility. In the experiments Tokuhiro, Kimura (1999) with interaction of a vertical non-buoyant jet and two parallel buoyant jets, such interaction (merging) is visible – even when the "hot" jets are separated with the "cold" one. That has one important implication: classic analyses of PTS with the REMIX codes taking into account only one cold plume could be non-conservative.

Other simulations

In 1997, preliminary announcement of Pressurized Thermal Shock International Comparative Study was released at OECD-NEA CSNI PWG-3 Intermediate Workshop in Paris, June 2-3, 1997. The problem statement was distributed in December 1996 and the term for submission of final results was October 1997. In the Task group THM (Thermal Hydraulic Mixing), a scenario with transient due to a 200 cm^2 leak in a hot leg of a 1300 MW 4 loop PWR was selected. The plant was fictitious, but some data from UPTF were adopted. Two tasks, Task PMIX (influence of different minimum downcomer water levels) and Task PINJ (influence of reduced emergency cooling water injection rate) were proposed. Distribution of water temperature and heat transfer coefficients in the downcomer was required. Only one CFD analysis was performed, that of Scheuerer (1998) who analysed the Task PINJ with TASCflow code. 180 000 cells were used with adiabatic outer walls and conjugate heat transfer model. Up to 4000s of the transient were calculated with an average time step size of 50s (8 iterations per step for convergence). No comparison with experiments was made in this scoping study, but some conclusions were formulated: buoyancy effects should be considered, and variable properties of water should be used.

A specific aspect of overcooling transients, oscillatory natural circulations during SB-LOCA overcooling transients in a PWR when cold water is injected into cold leg loop seals was tested in REWET-III facility, as described in Miettinen et al. (1987) and in Tuomisto (1987a).

Menant, Latrobe (2003) described an application of the TRIO-U CFD code to the computation of the transient flow in the real geometry of a 3 loop PWR. The part from the pump to core inlet was modelled with boundary conditions produced by CATHARE runs and a very detailed representation of the geometry (1.5 million nodes). Dynamic Smagorinsky SGS model was used, with 2nd order discretization in space, 3rd order discretization in time. The computation lasted 4500 hours on Compaq IXIA supercomputer, 20 processors were used in parallel. The computation had a character of a feasibility study, and no sensitivity study in the sense of the ECORA Best Practice guidelines could be performed.

In www.usnrc.gov/reading-rm/doc-collections/acrs/tr/subcommittee/2001/th010717.html, the website of the NRC, and th010718.html, there is very lengthy transcription of discussion which took place during an Advisory Committee on Reactor Safeguards, Thermal Hydraulic Phenomena Subcommittee Meeting in Corvallis, Oregon, US. The subject of this meeting was an overview of the Oregon State University Nuclear Reactor Research in the field of PTS. Both numerical (RELAP5, REMIX, STAR-CD) and experimental (APEX-CE) programs were discussed, including many visualisations. Next two references are in fact based on the discussed issues.

Haugh, Reyes (2001) applied STAR-CD computer code to CREARE one-half scale facility representing a 90°planar section of downcomer, core barrel, and lower plenum with cold leg, pump and loop seal. Only basic features of mixing after ECCS injection into the cold leg were studied. The solution domain does not correspond to the domain recommended by the Regional Mixing Model. The initial conditions were taken from the MAY 105 test with one stagnant loop, and three sensitivity calculations were performed to assess the effect of wall heat transfer. The benchmark indicated that the STAR-CD predicted well the type of mixing phenomena associated with PTS.

Yoon, Suh (1999) used the ANSYS-CFX code to analysis the effect of direct vessel injection on the Korean next generation reactor RPV shell temperature. Both steam and water in reactor vessel were considered for comparison. A similar computation is described by Matarazzo, Schwirian (1998).

Yoo, Jeon (2002) simulated four test cases with two or one jets flowing into a circular tube. The main goal of the tests was thermal striping (two parallel jets, cases A and B), but the cases C and D are suitable for PTS, since one jet flows into the tube either from below (case C) or from the top (case D). Three different RANS turbulence models were used: k-ε, l-k- ε, and full RSM model. The results were compared with simulations using the VLES (Very Large Eddy Simulation) approach. Since only limited measured data on the simulated cases are available, no definite conclusions have been formulated so far.

Boros, Aszodi (2002) performed a numerical analysis of coolant mixing in the downcomer of a VVER-440 type reactor with the code ANSYS-CFX-5.5.1.

Ref. 1: Armaly, B. F., Durst, F., Pereira, J. C. F., Schonung, B.: Experimental and theoretical investigation of backward-facing step flow. J. Fluid Mechanics 127 (1983) 473 – 496.

Ref. 2: Baughn, J. W., Shimizu, S. S.: Heat transfer measurements from a surface with uniform heat flux and a fully developed impinging jet. J. of Heat Transfer 111 (1989) 1096 – 1098.

Ref. 3: Boros, I., Aszodi, A.: Numerical analysis of coolant mixing in the RPV of VVER-440 type reactors with the code ANSYS-CFX-5.5.1. Technical Meeting on Use of Computational Fluid Dynamics (CFD) Codes for Safety Analysis of Reactor Systems, Including Containment. Pisa, Italy, 11-14 November 2002.

Ref. 4: Chen, C. J., Rodi, W.: Vertical turbulent buoyant jets. A review of experimental data. Pergamon Press 1980.

Ref. 5: Cooper, D., Jackson, D. C., Launder, B. E., Liao, G. X.: Impinging jet studies for turbulence model assessment. Part I: Flow-field experiments. Int. J. Heat Mass Transfer 36 (1993) 2675 – 2684.

Ref. 6: Dolan, F. X., Valenzuela, J. A.: Thermal and fluid mixing in ½-scale test facility. Vol. 1 – Facility and test design report. EPRI NP-3802, NUREG/CR-3426, September 1985.

Ref. 7: Egorov Y.: Validation of CFD codes with PTS-relevant test cases. ECORA deliverable 2004, ECORA web page http://domino.grs.de/ecora/ecora.nsf, Public Docs.

Ref. 8: Fanning, M. W., Rothe, P. H.: transient cooldown in a model cold leg and downcomer. EPRI NP-3118, May 1983.

Ref. 9: Freitas, C. J.: Perspective: Selected benchmarks from commercial CFD codes. Trans. ASME, J. Fluids Eng. 117 (1995) 208 – 218.

Ref. 10: Gango, P.: Application of numerical modelling for studying boron mixing in Loviisa NPP. OECD/CNSI Spec. Meeting on Boron Dilution Reactivity transients. State College PA USA, Oct. 18-20, 1995.

Ref. 11: Haugh, B., Reyes, J. N.: The use of STAR-CD to assess thermal fluid mixing in PWR geometry. Trans. ANS 85 (2001), 253 – 254.

Ref. 12: Hoehne T., Kliem S., Scheuerer M.: Experimental and Numerical Modelling of a Buoyancy-driven flow in a reactor pressure vessel. The 11th Int. Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11), Avignon, France, October 2-6, 2005. Paper 480.

Ref. 13: Hsu, J.-T., Ishii, M., Hibiki, T.: Experimental study on two-phase natural circulation and flow termination in a loop. Nucl. Eng. Design 186 (1998) 395 – 409.

Ref. 14: Iyer, K. N.: Thermal hydraulic mixing in the primary system of a pressurized water reactor during high pressure safety injection under stagnant loop conditions. PhD Thesis, Purdue University, December 1985.

Ref. 15: Iyer, K., Gherson, P., Theofanous, T. G.: PURDUE's one-half-scale high pressure injection mixing tests. Proc. 2nd Nuclear Thermal-Hydraulics Meeting of ANS, New Orleans, June 3-7, 1984, pp. 859-861.

Ref. 16: Iyer, K., Theofanous, T. G.: Decay of buoyancy driven stratified layers with applications to PTS: Reactor predictions. ANS Proc. 1985 National Heat Transfer Conf., Denver, CO, August 4-7, 1985, vol. 1, pp. 358. Nucl. Sci. Eng. 108 (1991), No. 2.

Ref. 17: Kotsovinos, N. E.: A study of the entrainment and turbulence in a plane buoyant jet. PhD thesis, California Institute of Technology, 1975.

Ref. 18: Lee, H., Moin, P.: Direct numerical simulation of turbulent flow over a backward-facing step. Stanford Univ. Center for Turbulence Research, Annual Research Briefs 1992, pp. 161 – 173.

Ref. 19: Maschek, W., Roth, A., Kirstahler, M., Meyer, L.: Simulation experiments for centralized liquid sloshing motions. Kernforschungszentrum Karlsruhe, report Nr. 5090, 1992.

Ref. 20: Matarazzo, J. C., Schwirian, R. E.: CFD analysis of a direct vessel injection (DVI) transient to calculate AP600 reactor vessel shell temperature. Proc. ASME Nuclear Engineering Division, NE vol. 22, 1-7, 1998.

Ref. 21: Menant, B., Latrobe, A.: LES interpretation of single phase PTS following the injection of under saturated water in the cold leg of a 3 loops PWR. Private communication, 2003.

Ref. 22: Miettinen, J., Kervinen, T., Tuomisto, H., Kanter, H.: Oscillations of single-phase natural circulation during overcooling transients. ANS Topical Meeting, Atlanta, April 12.15, 1987.

Ref. 23: Mustonen, P.: Fluid and thermal mixing tests of the Loviisa pressure vessel downcomer. Report IVO, Helsinki, April 1984.

Ref. 24: Pigny, S.: Description of selected test cases and physical models. Internal ECORA document, CEA-DRN-DTP, Grenoble 2002.

Ref. 25: Ransom, in Hewitt, G. F., Delhaye, J. M., Zuber, N. (eds.): Multiphase Science and Technology. 9 (1992) 591 – 609.

Ref. 26: Reyes, J. N., Groome, J. T., Lafi, A. Y., Franz, S. C., Rusher, C., Strohecker, M, Wachs, D., Colpo, S., Binney, S.: Final report of NRC AP600 research conducted at Oregon State University. US Nuclear Regulatory Commission, NUREG-CR-6641, August 1999.

Ref. 27: Reyes, J. N., Groome, J. T., Lafi, A. Y., Wachs, D., Ellis, C.: PTS thermal hydraulic testing in the OSU APEX facility. Int. J. Pressure Vessels and Piping 78 (2001), 185-196.

Ref. 28: Rohde, U. et al.: Validation of CFD Codes Based on Mixing Experiments. Final Report of the Work Package 4 of the FLOMIX-R Project, 2004.

Ref. 29: Rothe, P. H., Ackerson, M. F.: Fluid and thermal mixing in a model cold leg and downcomer with loop flow. EPRI NP-2312, April 1982.

Ref. 30: Rothe, P. H., Fanning, M. W.: Evaluation of thermal mixing data from a model cold leg and downcomer. EPRI NP-2773, December 1982.

Ref. 31: Rothe, P. H., Fanning, M. W.: Thermal mixing in a model cold leg and downcomer at low flow rates. EPRI NP-2935, March 1983.

Ref. 32: Rothe, P. H., Marscher, W. D.: Fluid and thermal mixing in a model cold leg and downcomer with vent valve flow. EPRI NP-2227, March 1982.

Ref. 33: Sarkar, J., Liebert, J.: UPTF test instrumentation; measurement system identification, engineering units and computed parameters. KWU Work Report R515/85/23, Erlangen, September 13, 1985.

Ref. 34: Scheuerer, M.: Reactor Pressure Vessel – International Comparative Assessment Study RPV ICAS. Analyses on Thermal Hydraulics Mixing (THM) Tasks. Technical Note. Workshop on the CSNI Project RPV ICAS, February 25 – 27, 1998, Orlando, Florida, USA.

Ref. 35: Scheuerer, M.: International Comparative Assessment Study of Pressurized-Thermal-Shock: Task Group THM, Parametric Study PINJ. Report.

Ref. 36: Scheuerer, M.: Selection of PTS-relevant test cases. Internal ECORA document D05, 2002. ECORA web page http://domino.grs.de/ecora/ecora.nsf, Public Docs.

Ref. 37: Simpson, A. R.: Large water-hammer pressures for column separation in pipelines. J. of Hydraulic engineering 117 (1989) 1310 – 1316.

Ref. 38: Tenhumberg, M., Wenzel, H.-H.: Verzuchsprotokoll Temperaturschichtversuche im RDB Versuchsgruppe TEMP Hauptuersuche T32.11-91. PHDR-Arbeitsbericht No. 3.468/85, KfK GmbH, August 1985.

Ref. 39: Theofanous, T. G., Yan, H. A.: A unified interpretation of one-fifth to full scale thermal mixing experiments related to pressurized thermal shock. NUREG/CR-5677 (1991).

Ref. 40: Theofanous, T. G., Nourbakhsh, H. P., Gherson, P., Iyer, K.: Decay of buoyancy-driven stratified layers with applications to pressurized thermal shock. NUREG/CR-3700, May 1984.

Ref. 41: Theofanous, T. G., Iyer, K., Nourbakhsh, H. P., Gherson, P.: Buoyancy effects in overcooling transients calculated for the NRC pressurized thermal shock study. NUREG/CR-3702, May 1986.

Ref. 42: Theofanous, T. G., Gherson, P., Nourbakhsh, H. P., Iyer, K.: Decay of buoyancy-driven stratified layers with applications to pressurized thermal shock. Part II: PURDUE's ½ scale experiments. NUREG/CR-3700, May 1984. Nucl. Eng. Des. 1991.

Ref. 43: Theofanous, T. G., Angelini, S., Yan, H.: Universal treatment of plumes and stresses for pressurized thermal shock evaluations. NUREG/CR-5854, June 1992.

Ref. 44: Tokuhiro, A., Kimura, N.: An experimental investigation on thermal striping mixing phenomena of a vertical non-buoyant jet with two adjacent buoyant jets as measured by ultrasound Doppler Velocimetry. Nucl. Eng. Design 188 (1999) 49 – 73.

Ref. 45: Toppila, T.: Experience with validation of CFD methods for pressure vessel downcomer mixing analyses. Technical Meeting on Use of Computational Fluid Dynamics (CFD) Codes for Safety Analysis of Reactor Systems, Including Containment. Pisa, Italy, 11-14 November 2002.

Ref. 46: Toppila, T.: Selected experiments at the Fortum PTS test facility. FLOMIX-R 2nd Project Meeting, Älvkarleby, Sweden, April 22-23, 2002.

Ref. 47: Tuomisto, H., Mustonen, P.: Thermal mixing tests in a semiannular downcomer with interacting flows from cold legs. Test Report RLB-340, IVO, Helsinki, May 1986.

Ref. 48: Tuomisto, H., Mustonen, P.: Thermal mixing tests in a semiannular downcomer with interacting flows from cold legs. NUREG/IA-0004, October 1986.

Ref. 49: Tuomisto, H.: Thermal mixing tests in a semiannular downcomer with interacting flows from cold legs. Proc. 14th Water Reactor Safety Information Meeting, Gaithersburg, MD, October 27-31, 1986, vol. 4, pp. 341-361.

Ref. 50: Tuomisto, H.: Experiments and analysis of thermal mixing and stratification during overcooling accidents in a pressurized water reactor. ANS Proceedings 1987 National Heat Transfer Conference, Pittsburgh, PA, August 9-12, 1987.

Ref. 51: Tuomisto, H.: Thermal-hydraulics of the LOVIISA reactor pressure vessel overcooling transients. IVO-A-01/87, Helsinki 1987.

Ref. 52: Valenzuela, J. A., Dolan, F. X.: Thermal and fluid mixing in ½-scale test facility. Vol. 2 – Data report. EPRI NP-3802, NUREG/CR-3426, September 1985.

Ref. 53: Weiss, P. A.: UPTF experiment operating specification of Test 1. KWU R515/Ws-et, Erlangen, April 2, 1986.

Ref. 54: Weiss, P. A.: Fluid-fluid mixing test; A quick look at the essential results. KWU R515, 2D3D Analysis Meeting, Erlangen, June 5-13, 1986.

Ref. 55: Weiss, P. A. et al.: Fluid-fluid mixing test; Quick look report. KWU R515/87/1, Erlangen, January 1987.

Ref. 56: Weiss, P. A. et al.: Fluid-fluid mixing test; Experimental data report. KWU R515/87/09, Erlangen, April 1987.

Ref. 57: Willemsen S. M., Komen Ed M. J.: Assessment of RANS CFD modelling for Pressurized Thermal Shock analysis. The 11th Int. Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11), Avignon, France, October 2-6, 2005. Paper 121.

Ref. 58: Wolf, L. et al.: Extracts of the design report 3.150/84 for thermal mixing experiments in cold leg and downcomer. HDR-Test Group TEMB T32, KfK GmbH, November 1984.

Ref. 59: Wolf L., Schygulla U., Haefner W., Fischer K., Baumann W.: Results of thermal mixing tests at the HDR-facility and comparisons with best-estimate and simple codes. Nucl. Eng. Design 99 (1987), 287-304.

Ref. 60: Wolf L., Haefner W., Fischer K., Schygulla U., Baumann W.: Application of engineering and multidimensional finite difference codes to HDR thermal mixing experiments TEMB. Nucl. Eng. Design 108 (1988), 137-165.

Ref. 61: Wolf, L., Schygulla, U.: Comparison between data and blind pre- and post-test calculations for the three preliminary thermal mixing tests at the HDR facility; HDR-TEMB Experiments T32.15, T32.17 and T32.18. PHDR Internal Working Report No. 3.452/85, KfK GmbH, April 1985.

Ref. 62: Yoo G. J., Jeon W. D.: Analysis of unsteady turbulent merging jet flows with temperature difference. ICONE10-22235, Proceedings of ICONE10, 10th Int. Conf. On Nucl. Eng., Arlington, VA, April 14-18, 2002.

Ref. 63: Yoon, S. H., Suh, K. Y.: Analysis of direct vessel injection flow pattern using the ANSYS-CFX code. Trans. ANS 81(1999) 334-335.

Ref. 64: Young, E. P., Reyes, J. N.: A comparative analysis of APEX-CE and STAR-CD of fluid mixing in the cold leg and downcomer of a PWR. Trans. ANS 85 (2001), 258 – 259.

5.3 Thermal Fatigue

Failures of parts of structures of NPPs caused by thermal fatigue have been recorded for Genkai Unit 1 (JP), Tihange Unit 1 (BE), Farley Unit 2 (US), Phénix (FR), PFR (UK), Tsuruga Unit 2 (JP) and Loviisa (FI). Consequently, considerable effort has been devoted to research of the phenomenon, and both experimental and numerical information is being gathered to aid understanding.

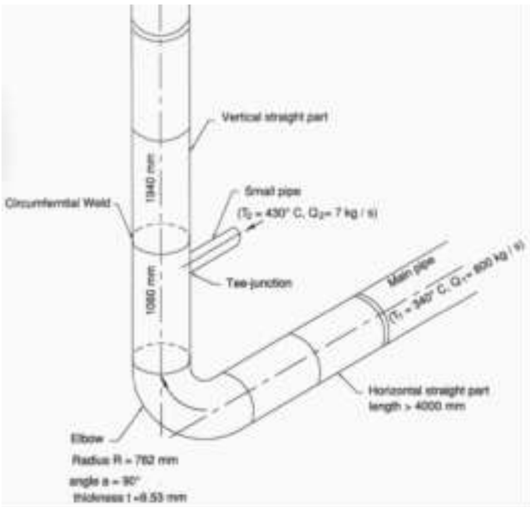
Thermal fatigue (thermal striping) is studied mainly for two geometric configurations: (1) T-junctions, and (2) for two or more parallel jets in contact with neighbouring structures. The problem is complex, since it involves several scientific disciplines and, consequently, several computer codes: computation of velocity and temperature fields in flowing fluids, computation of temperature fields in solids, computation of mechanical stresses in solids, and computation of behaviour of cracks in solids. Any experimental database should reflect and comprehensively cover all these fields of discipline. Moreover, coupling between the fields could be two-way, which means computations have to be carried out simultaneously, the data from each being appropriately interfaced.

T-junctions

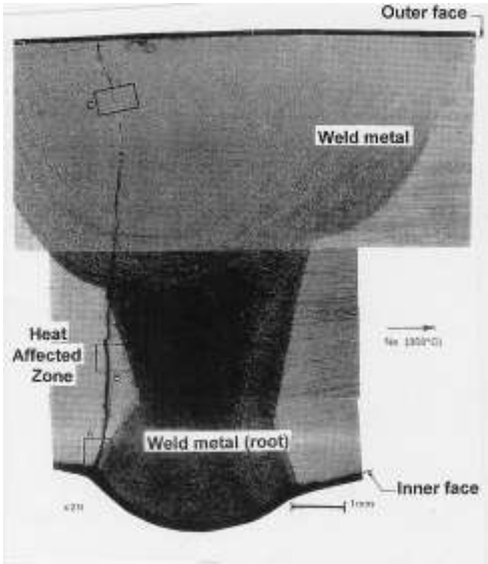
Liquid-Metal Reactors

The Phénix 300 MW(e) prototype reactor is a sodium-cooled fast breeder reactor. As a liquid metal, sodium has a high thermal conductivity. This, combined with a large temperature difference (core inlet / outlet = 400 / 550 °C) and highly turbulent flow conditions, leads to a potential thermal striping problem. Early in the design process, this risk had been taken into account by installing static mixers in some of the T-junctions of the secondary loops. In addition, local temperature measurements were taken in-situ in some stratified or mixing zones with the reactor online.

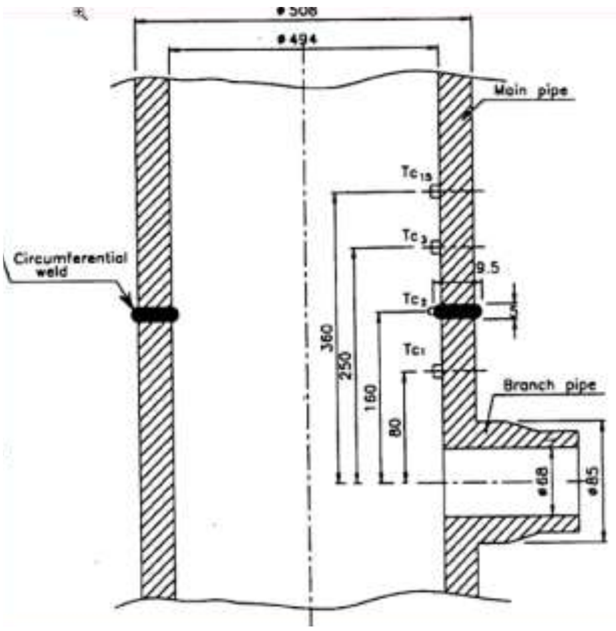
Despite these precautions, in the 1990s, a crack was detected at a T-junction between a small pipe (carrying hot sodium from the hydrogen detection device) and the main secondary loop (cold branch). A sketch of the configuration is given in the Figure below.



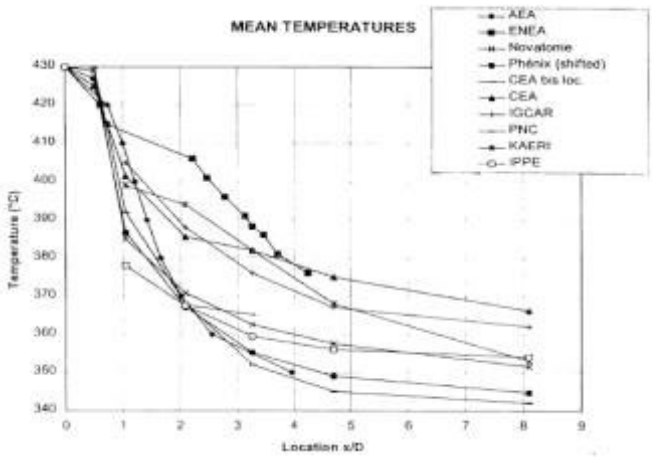
The pipe was cut off and replaced, the original section then being analysed from a metallurgical standpoint. Visual inspections of the cut piece revealed the shape of the thermal peak loading region on the main branch pipe. In this configuration and for the given flow rates, the hot flow from the branch line does not penetrate the main stream, but is deflected along the near surface of the cold pipe wall, and oscillates azimuthally. Moreover, a slight swirl flow created by the pipe bends immediately upstream in the cold branch leads to deviations of the thermally striped zone. The Figure below shows details of the crack detected.



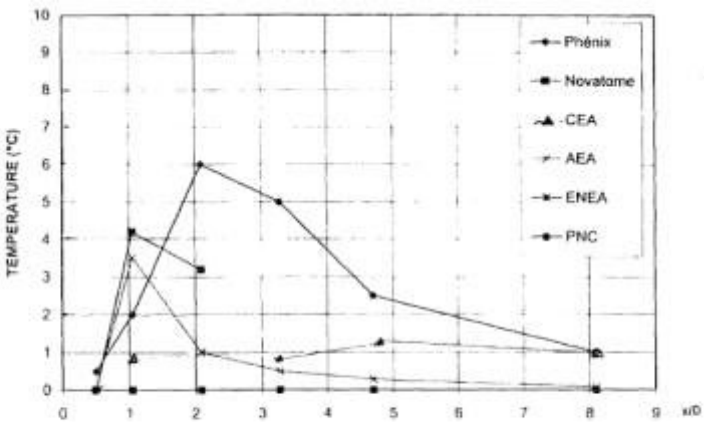
Temperature measurements were taken for the operating loop (Ref. 4). Thermocouples were located on the pipe outer surface at 15 locations: 4 along the meridian line downstream of the junction on the hot side, 2 at the junction, 4 around the circumference away from the meridian line; and 2 at 180° (i.e., on the opposite wall) from the meridian line. Data acquisition intervals were 1 ms for the short record, and 1.5 s for the long record. Temperature records showed a slight skew-symmetry of the temperature distribution, indicating that the jet from the branch pipe had been directed sideways. Instantaneous temperatures were recorded for each thermocouple over a time period of about 2000 seconds. The Figure below shows the locations of the thermocouples.



The maximum linearised temperature difference across the wall is about 12K, with a non-linear peak component of 2K. These estimates were obtained after reconstituting the temperatures on the inner wall surface from the measured values and their associated frequencies. The maximum achievable frequency is about 0.25 Hz; higher frequencies than this are not observable. The two Figures below show the experimental results in terms of average temperatures and thermal fluctuations along the meridian line versus the distance from the junction (Ref. 4).

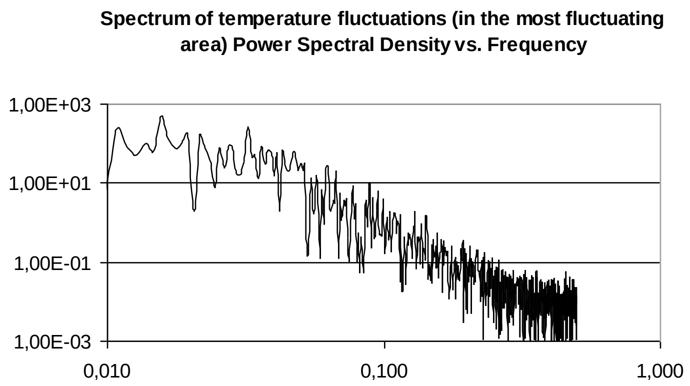


Average Temperature



Fluctuations of Temperature

The spectrum of temperature fluctuations, in the most fluctuating area, is plotted in the Figure below.



In the context of the international benchmark exercise sponsored by the IAEA in the 1990s, combined CFD, stress analysis and fatigue calculations have been performed by several international teams, conclusions from which are given in Ref. 4. As well as these tests, specific experiments on a scale model T-junction in sodium were performed in the 1980s (the CASTOR tests). Here, a moving rake of thermocouples, located downstream the tee provided average values and fluctuations of temperature. Some details are given in Ref. 16.

Thermal striping was the subject of benchmark studies performed within the co-ordinated research project *Harmonization and Validation of Fast Reactor Thermomechanical and Thermo-Hydraulic Codes and Relations using Experimental Data*. A benchmark exercise on "T-junction of LMFBR secondary circuit" was approved, representing the secondary circuit of the French Phénix LMFBR. A set of experimental data was made available to the participating institutes. The CFD codes Trio-VF, STAR-CD, AQUA, DINUS-3, PHOENICS and CFX-4 were used in the exercise. In the recommendations, application of the pseudo-direct Navier-Stokes simulation is mentioned (LES without SGS models) as a possibility, but full LES is recommended. Application of RANS models requires a priori assumptions regarding the frequencies, and the range of the frequencies considered damaging for a particular pipe wall thickness must be determined in advance. Frequencies lower than this band do not produce a sufficient ΔT across the wall, and higher frequencies cannot penetrate the wall. The physical time of calculation had to cover at least 10 periods of the lower band of frequency, and the time step of the computation chosen in order to be able to capture the upper bound of frequency. The boundary conditions should include secondary flows (e.g. swirl flow) and low frequency variations of temperature and/or velocity.

Light Water Reactors

Nakamori et al. (1998) describe Japanese tests to investigate mixing behaviour of leak flow with stagnant fluid in a branch pipe downstream of a check valve. The branch pipe was made of transparent acrylic and connected to the simulated main coolant pipe. The leak-simulated fluid was coloured to observe the mixing phenomena and contained 30% CaCl for simulating the density difference between the high temperature main coolant and the low temperature leak fluid. The test conditions are detailed in the Table below.

Test cases	Type of branch	Pressure [MPa]	Hot water temperature in the main coolant pipe4 [K]	Main coolant pipe velocity [m/s]	Leak flow temperature [K]	Leak flow rate [kg/h]
Small leak test	Type 1, 2	15.49	563, 596	5.5, 16	290-300	10
Large leak test	Type 2	15.49	596	16	290-300	30-300

The Type 1 branch is horizontal; the Type 2 branch is vertically downwards. Temperature measurements were taken at 24 axial locations for the Type 1 branch, and at 8 axial locations for the Type 2 branch.

Thermal fatigue in T-junctions has also been studied within the EU 5th FWP project THERFAT (Thermal Fatigue Evaluation of Piping System Tee-connections”). Within the project, thermal-hydraulic tests were carried out to simulate, illustrate, measure and quantify the turbulent fluid flow and associated thermal loads in various mixing tee configurations. The tests cover:

- visualisation of the turbulent fluid phenomena in glass models,
- electrical conductivity measurements in glass models simulating the temperature differences by using salt water with different specific densities at ambient temperature,
- measurement of the temperature fluctuation spectra occurring in steel models with test temperature differences up to 90K.

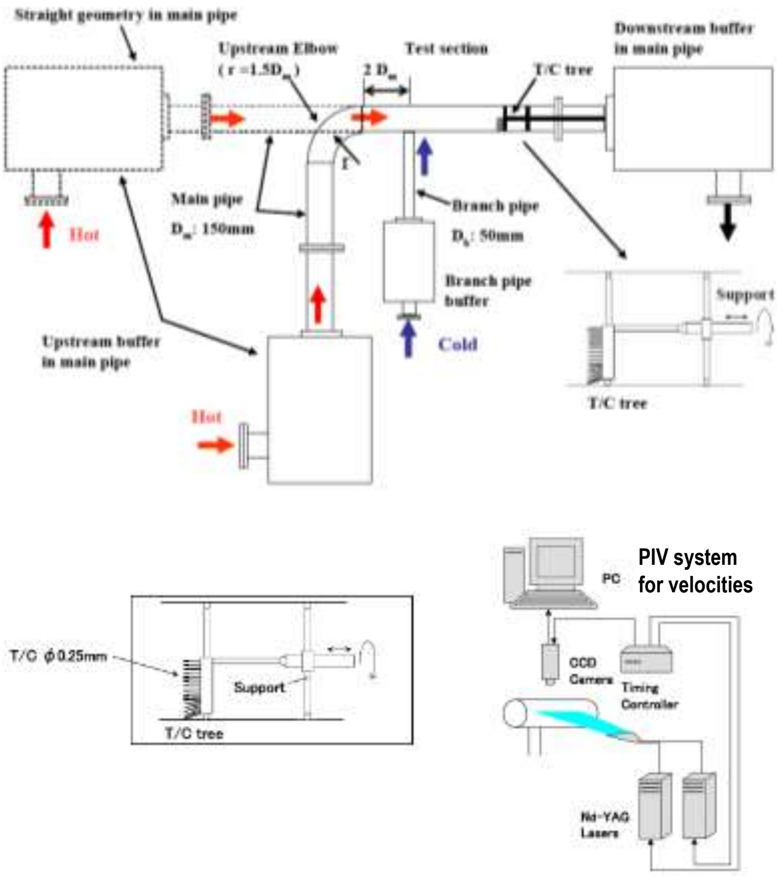
The following tee configurations were selected for the thermo-hydraulic tests:

- DN 50:50 mm tee: perpendicular branch in different configurations, glass and steel models;
- DN 75:25 mm tee: perpendicular branch in different configurations, glass and steel models;
- DN 50:50 tee: 45° branch in different configurations, glass model for visualisation only;
- DN 100:100 mm tee: perpendicular branch, glass model.

The test with the DN 50:50 mm perpendicular branch was subsequently analysed using CFD codes using the classical k-ε and LES turbulence modelling approaches. The determination of fluid-to-wall heat transfer coefficients was the main focus of these computations. Only the LES approach was shown to be able to reproduce the turbulent temperature fluctuations observed in the tests, though the k-ε formulation was shown to be able to simulate those cases in which low-frequency thermal fluctuations are produced due to non-convected, large-scale instabilities, such as those associated with pulses, pump fluctuations, gravity waves, etc. Good agreement of computed and measured results was found, but long computational times were needed, especially for the LES simulations.

Experiments have been carried out at the Long Cycle Fluctuation (WATLON) facility, O-arai Engineering Center, Japan. Water was the working fluid. The geometry tested represents a horizontal pipe with an upstream elbow of diameter 150 mm in the vertical plane, and a T-junction of diameter 50 mm in the same plane from below. The test section is made of transparent acrylic. The flow velocity was 0.1 m/s to 3.0 m/s in the main pipe and 0.5 m/s to 2 m/s in the branch pipe. The temperature difference was zero (isothermal conditions). An Ar laser light sheet was used to visualise the flow patterns in one cross-section of the T-junction, and a thermocouple tree was used to measure the fluid temperature inside the main pipe. The tree could be rotated circumferentially, and also moved in the axial direction. High-speed Particle Imaging Velocimetry (PIV) was applied to measure the flow velocity distribution in the tee.

A unique feature of these tests was that it was possible to compare the effect of an upstream elbow on the mixing at the T-junction against that for a straight pipe.



(a)

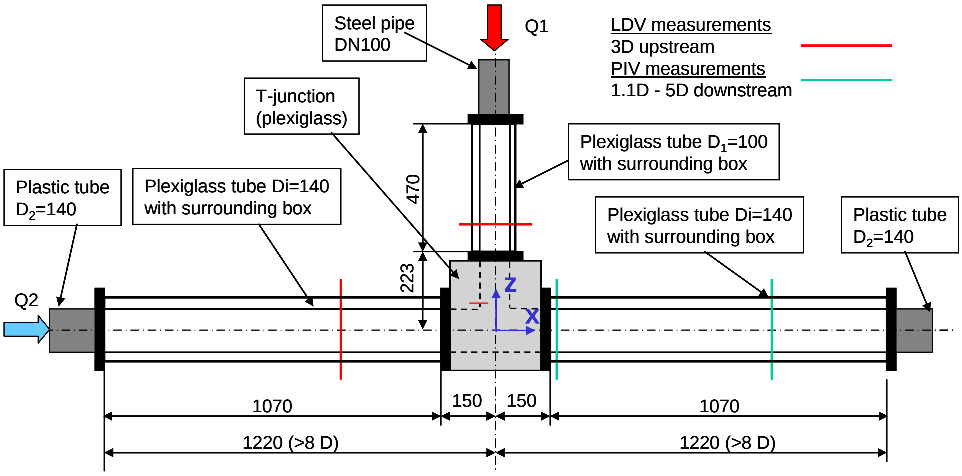
The WATLON experimental facility: layout; thermocouple rake; and PIV system.

Three test cases with different flow combinations were performed:

	Flow Pattern	Velocity in the main pipe [m/s]	Velocity in the branch pipe [m/s]	Momentum ratio (main/branch pipe)
Case 1	Wall jet	1.46	1.0	8.1
Case 2	Deflecting jet	0.46	1.0	0.8
Case 3	Impinging jet	0-23	1.0	0.2

Time-averaged velocities and temperatures, and their fluctuation intensities, at various positions in the main pipe were provided for all cases. Dangerous frequency components around 6 Hz, or even lower, were found in all cases. A kind of Karman vortex street behind the branch pipe jet appeared, which could be the cause. Also, it was noted that the presence of the elbow could cause disturbances leading to low frequency (less than 5 Hz) fluctuations.

Numerical simulations of flow in a mixing tee using the LES model of turbulence can be found for the Civaux Unit 1 case, employing the thermal-hydraulic/thermo-mechanical computer code CAST3M. Calculations have also been performed using the thermal-hydraulic code Saturne (FVM), coupled to the conjugate heat transfer module Syrthes (FEM). In addition, FLUENT simulations have been carried out for the Hitachi co-current experiment (one inlet in branch pipe, one inlet in main pipe, outlet in main pipe) and the Toshiba collision-type experiment (both inlets in the main pipe, outlet in branch pipe).



T-junction test section showing LDV and PIV measurement stations

As part of an ongoing commitment to extend the assessment database for the application of CFD to nuclear reactor safety issues, the Special CFD Group within the scope of activities of the OECD/NEA Working Group on the Analysis and Management of Accidents (WGAMA) launched an blind international numerical benchmarking exercise based on a T-junction experiment performed at the Älvkarleby Laboratory of Vattenfall Research and Development in Sweden.

A date was fixed for the kick-off meeting for the benchmark exercise (20 May, 2009). An announcement was prepared, with an invitation to register interest in receiving the benchmark specifications. Of the 750 or so recipients of this invitation, 65 registrations were received from organisations in 22 countries, of whom 28 attended the kick-off meeting.

A draft version of the specifications was circulated to all registered participants on June 30, 2009 with an invitation for feedback concerning errors, clarity, ambiguity and possible misunderstandings. With very few changes, the final and official version was circulated on July 15, 2009. This gave participants 9½ months to complete their calculations and submit their results by the deadline date of April 30, 2010. In total, 29 were received by this date. These formed the basis of a thorough synthesis of the results.

Full details are given in Refs. 21, 22.

Parallel jets

Kimura et al. (2005) describe sodium and water experiments with parallel triple jet flow along a wall. Unstable behaviour of the jets leads to temperature fluctuations in the wall, which could cause thermal fatigue. The cases tested (cold central jet with hot side jets) are presented in the Table below.

Flow pattern	Fluid	Case Name	Hot Jets			Cold Jet			Average	
			V (m/s)	Re x10^4	T (°C)	V (m/s)	Rex10^4	T (°C)	ΔT (°C)	V_{av}(m/s)
Isovelocity	Water	WE3	0.49	1.47	39.3	0.52	1.25	28.5	10.8	0.50
	Sodium	SE3-V	0.51	2.82	347.5	0.51	2.60	304.5	43.0	0.51
		SE3-R	0.30	1.67	349.9	0.30	1.55	310.0	39.9	0.30
Non-isovelocity	Water	WN3	0.49	1.47	39.3	0.34	0.79	26.2	13.1	0.44
	Sodium	SN3-V	0.51	2.87	349.8	0.32	1.68	311.0	38.8	0.45
		SN3-R	0.31	1.71	352.3	0.20	1.04	311.0	41.3	0.27

Experiments with a vertical non-buoyant jet with two adjacent buoyant jets have also been carried out. Another Japanese experiment with two jets of hot and cold water has been simulated with STAR-CD using an LES model of turbulence. In the experiment, vertical hot (46°C) and cold (15°C) jets of water with velocity 3.36 m/s impinge on a test piece placed above. Main frequencies of the thermal fluctuations were 7.5 Hz in the calculations and 5–7 Hz in the experiment.

Computational analysis of two test cases with parallel jets and two test cases with one jet flowing into a circular tube is also available. An approach combining steady RANS, in order to identify possible regions of strong thermal striping, and "pseudo-DNS", used earlier is replaced here with an LES (or more precisely a VLES) approach.

Ref. 1: F. Archambeau, N. Méchitoua, M. Sakiz: "Code_Saturne: a Finite Volume Code for the Computation of Turbulent Incompressible Flows – Industrial Applications", Int. J. on Finite Volumes, **11**, 2-62 (2001).

Ref. 2: S. Chapuliot, C. Gourdin, T. Payen, J.-P. Magnaud, A. Monavon: Hydro-thermal-mechanical analysis of thermal fatigue in a mixing tee, Nucl. Eng. Des., **235**, 575-596 (2005).

Ref. 3: S.-K. Choi, M.-H. Wi, W.-D. Jeon, S.-O. Kim, "Computational study of thermal striping in an upper plenum of KALIMER", Nucl. Technology **152**, 223-238 (2005).

Ref. 4: O. Gélineau, M. Spérandio, J.-P. Simoneau, J.-M. Hamy, P. Roubin, 2002, "Validation of fast reactor thermomechanical and thermohydraulic codes : thermomechanical and thermal hydraulic analyses of a tee junction using experimental data", Final report of a co-ordinated research project, International Atomic Energy Agency, AIEA TECDOC-1318, Nov. 2002.

Ref. 5: O. Gélineau, C. Escaravage, J.-P. Simoneau, C. Faidy "High Cycle Thermal Fatigue: Experience and State of the Art in French LMFR, Proc. SMIRT16, 2001.

Ref. 6: L.-W. Hu, M.S. Kazimi, “Large Eddy Simulation of Water Coolant Thermal Striping in a Mixing Tee Junction”, NURETH-10, Seoul, Korea, Oct. 5-9, 2003.

Ref. 7: L.-W. Hu, M.S. Kazimi, “LES benchmark study of high cycle temperature fluctuations caused by thermal striping in a mixing tee”, Int. J. Heat and Fluid Flow, **27**, 54-64 (2006).

Ref. 8: N. Kimura, M. Nishimura, H. Kamide, “Study on convective mixing for thermal striping phenomena – experimental analyses on mixing process in parallel triple-jet and comparisons between numerical methods”, ICONE-9, 2001.

Ref. 9: N. Kimura, H. Miyakoshi, H. Ogawa, H. Kamide, Y. Miyake, K. Nagasawa, “Study on convective mixing phenomena in parallel triple-jet along wall – comparison of temperature fluctuation characteristics between sodium and water”, NURETH-11, Paper 427, 2005.

Ref. 10: K.-J. Metzner, U. Wilke, “European THERFAT project – thermal fatigue evaluation of piping system Tee – connections”, Nucl. Eng. Des., **235**, 473-484 (2005).

Ref. 11: T. Muramatsu, “Numerical analysis of non-stationary thermal response characteristics for a fluid-structure interaction system”, Trans. ASME (J. Pressure Vessel Technol.), **121**, 276–282 (1999).

Ref. 12: N. Nakamori, K. Hanzawa, K. Oketani, T. Ueno, J. Kasahara, S. Shirahama, “Research on thermal stratification in unisolable piping of reactor coolant pressure boundary”, Proc. Specialists Meeting on Experience with Thermal Fatigue in LWR Piping Caused by Mixing and Stratification, Paris, France, 8-10 June 1998. NEA/CSNI/R(98)8, pp. 229-240. http://www.oecdnea.org/html/nsd/docs/1998/csni-r98-8.pdf.

Ref. 13: H. Ogawa H., M. Igarashi, N. Kimura, H. Kamide, “Experimental study on fluid mixing phenomena in T-pipe junction with upstream elbow”, NURETH-11, Paper 448, 2005.

Ref. 14: Ch. Péniguel, M. Sakiz, S. Benhamadouche, J.-M. Stephan, C. Vindelrinho, “Presentation of a numerical 3D approach to tackle thermal striping in a PWR nuclear T-Junction”, PVP-Vol. 469, Design and Analysis of Pressure Vessels and Piping: Implementation of ASME B31, Fatigue, ASME Section VIII, and Buckling Analyses. PVP2003-2191. ASME 2003.

Ref. 15: J.-P. Simoneau, O. Gelineau, “Simulation of attenuation of thermal fluctuations near a plate impinged by jets”, ICONE-9, 2001.

Ref. 16: J.-P. Simoneau H. Noé, B. Menant, “Large eddy simulation of sodium flow in a tee junction, comparison of temperature fluctuations with experiments”, Proc. 8th Topical Mtg. Nuclear Reactor Thermal Hydraulics (NURETH-8), Kyoto, Japan, 1997.

Ref. 17: H.G. Sonnenburg, “Thermal Stratification in Horizontal Pipes Investigated in UPTF-TRAM and HDR Facilities”, Proc. Specialists Meeting on Experience with Thermal Fatigue in LWR Piping Caused by Mixing and Stratification, Paris, France, 8-10 June 1998. NEA/CSNI/R(98)8, pp. 201-228. http://www.oecdnea.org/html/nsd/docs/1998/csni-r98-8.pdf.

Ref. 18: Validation of fast reactor thermomechanical and thermohydraulic codes. Final report of a co-ordinated research project 1996-1999, IAEA-TECDOC-1318, IAEA, Nov. 2002.

Ref. 19: J. Westin et al., “Experiments and Unsteady CFD Calculations of Thermal Mixing in a T-Junction”, Proc. Int. Workshop on Benchmarking of CFD Codes for Application to Nuclear Reactor Safety (CFD4NRS), Garching, Munich, Germany, 5-7 September 2006 (CD-ROM).

Ref. 20: R. Zboray et al., “Investigations on mixing phenomena in sigle-phase flows in a T-junction geometry”, Proc. 12th Int. Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-12) Pittsburgh, Pennsylvania, U.S.A., September 30-October 4, 2007.

Ref. 21: Report of the OECD/NEA—Vattenfall T-Junction Benchmark Exercise, OECD Nuclear Energy Agency report, NEA/CSNI/R(2011)5, May 2011.

Ref. 22: B.L. Smith, J.H. Mahaffy, K. Angele, “A CFD benchmarking exercise based on flow mixing in a T-Junction”, Paper 145, 14th Int. Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-14), Toronto, Canada, Sept. 25-30, 2011.

5.4 Aerosol Transport in Containments

Despite that (based on PHEBUS experimental results), …“there is no indication that detailed CFD models are needed to calculate the global behaviour (of aerosols)”…, see Section 3.18 of this report, CFD codes could make a substantial contribution to the development of models or semi-empirical correlations to be used for the formation, transport and deposition of aerosols in NPP circuits. The models and correlations can then be used in less-detailed, lumped parameter codes. However, the detailed CFD approach could bring better understanding of physical processes taking place during experiments involving aerosol behaviour. It is therefore desirable to assess CFD codes also for this kind of application. Moreover, the conclusions reached for the highly idealised PHEBUS containment geometry may not extrapolate to the complex geometries of actual containments.

A possible experimental database could include former OECD/NEA activities in the field of aerosol behaviour: ISP-37 (VANAM M3 Aerosol behaviour in the Battelle Model Containment), the AHMED Code Comparison Exercise, ISP-44 (KAEVER test facility, VTT, Finland), and CEC benchmark problems. However, the most cited reference remains the Phebus FP Severe Accident Experimental Program, in which aerosol size distribution and composition, and interaction between vapours and aerosols are among the outcomes of the experiments. These activities focused primarily on lumped parameter codes, but CFD codes were used within Work Package 2 of the PHEBEN2 EU-supported project, based on the PHEBUS FPT0 and FPT1 experiments. The aim of this WP was “…less to validate the codes themselves than to understand the phenomena involved, and their quantitative contribution to the observed results.” It was found that the coupling between the thermal-hydraulics and the aerosol physics in the PHEBUS containment is 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 as functions of time…” CFD codes CFX 4.3, CFX 5.7 (FPT1 only) and TRIO VF were used. There were problems with comparison of measured values with calculated ones since “…only a few internal temperature measurements and no velocity measurements are available from PHEBUS.” Comparison with computation of FPT1 by means of the MELCOR 1.8.5 lumped parameter code was also made.

In Finland, aerosol behaviour is studied in the HORIZON facility, which is a scaled-down model of VVER-440 steam generator, and in the VICTORIA multi-compartment test facility, which is a scaled–down model of the containment of the Loviisa NPP. For this test, some experimental results were shown alongside CFD simulations using the FLUENT computer code.

A multi-level simulation of aerosol dynamics after sodium combustion is described in Yamaguchi et al. (2002). A set of tools is used including AQUA-SF CFD computer code. References on corresponding experiments lead mostly to documents in Japanese. One of the computer codes of the described set, SPHINCS for simulation of sodium fires on the largest scale was validated using experiments.

In summary, though there seems to be a consensus of opinion that aerosol deposition in containments is a high priority one for NRS, and that CFD has the potential to bring better predictions of aerosol deposition, the case for CFD playing an essential analysis role appears not to be proven. In any event, there is a clear lack of validation data for CFD models for this topic.

Ref. 1: Auvinen A., et al., Severe accident aerosol research in Finland. Proc. 3rd Finnish-French colloquium on Nuclear Power Plant safety, June 27-28, 2000, Lappeenranta, Finland.

Ref. 2: Clément B., et al., LWR severe accident simulation: synthesis of the results and interpretation of the first Phebus FP experiment FPT0. Nucl. Eng. Design, 226, 5-82 (2003).

Ref. 3: Firnhaber M., Kanzleiter T. F., Schwarz S., Weber G.: International Standard problem ISP37. VANAM M3 – A multi compartment aerosol depletion test with hygroscopic aerosol mterial. Comparison Report OCDE/GD(97)16, December 1996.

Ref. 4: Firnhaber M., Fischer K., Schwarz S., Weber G.: International Standard Problem ISP-44 KAEVER. Experiments on the behavior of core-melt aerosols in a LWR containment. NEA/CSNI/R(2003)5.

Ref. 5: Fischer K., Schall M., Wolf L.: CEC Thermal Hydraulic Benchmark Exercise on Fiploc Verification Experimental Phases 2, 3 and 4 – Results of Comparisons. EUR 14454 EN, 1993.

Ref. 6: Futugami S. et al. Pool combustion behavior of liquid sodium. Proc. 36th Japanese Symposium on Combustion, D311, 1998 (in Japanese).

Ref. 7: Gauvain J.: Post-test calculations of thermal hydraulic behaviour in DEMONA experiment B3 with various computer codes used in EC member states. EUR 12197 EN, 1989.

Ref. 8: Jones A. V., et al., Validation of severe accident codes against Phebus FP for plant applications: Status of the PHEBEN2 project, Nucl. Eng. Design, 221, 225-240 (2003).

Ref. 9: Ludwig W., Brown C. P., Jokiniemi J. K., Gamble R. E. CFD simulation of aerosol deposition in a single tube of a passive containment condenser, ICONE-9, 2001.

Ref. 10: Makynen J., Jokiniemi J. (eds.): CSNI/PWG4/FPC AHMED Code Comparison Exercise. NEA/CSNI/R(95)23, October 1995.

Ref. 11: Martín-Fuertes F., Barbero R., Martín-Valdepenas J. M., Jiménez M. A. Analysis of source term aspects in the experiment Phebus FPT1 with the MELCOR and CFX codes, Nucl. Eng. Des., 237, 509-523 (2007).

Ref. 12: Von der Hardt P., Jones A. V., Lecomte C., Tattegrain A. Nuclear Safety Research: The Phebus FP Severe Accident Experimental Program, Nucl. Safety, 35, 187-205 (1994).

Ref. 13: Yamaguchi A., Tajima Y. Validation study of computer code SPHINCS for sodium fire safety evaluation of fast reactor, Nucl. Eng. Des., 219, 19-34 (2003).

5.5 Sump Clogging

In 1992, a safety relief valve inadvertently opened on a steam line at the Barsebäck-2 BWR nuclear plant in Sweden. The steam jet stripped fibrous insulation from the adjacent piping systems. Part of the insulation debris was transported to the wetwell pool, and this debris subsequently clogged the intact strainers of the drywell spray system about 1 h after the start of the incident. Although the event in itself was not serious, it revealed a weakness in the defence-in-depth strategy of the plant, which under other circumstances could have led to the emergency core cooling system (ECCS) failing to provide recirculation water to the core. A similar incident occurred twice in 1993 at the Perry NPP in Ohio, USA.

Research and development efforts of varying degrees of intensity have been launched in many countries as a consequence. The corresponding knowledge bases have been updated several times, and workshops on the subject have also been organised. The international activities have been summarised in a NUREG report of the US NRC, which includes a model of fibre release under the influence of a jet, an empirical equation for the difference in pressure across the sieve as a function of fibre load, and the respective results of specifically designed material loadings experiments. All these activities reflect, in most cases, the views of the regulators and utilities. In parallel, efforts to investigate the problem in more detail from a mechanistic standpoint, particularly with the aim of CFD model development, are also being pursued.

As a result of these incidents, knowledge of insulation debris generation and transport is gaining in importance in regard to reactor safety research for both PWRs and BWRs. The insulation debris released near the break consists of a mixture of fibres and particles of very different sizes, shapes and consistency. Experiments have been performed at the University of Applied Science, Zittau/Görlitz in Germany in which original samples of mineral wool insulation material have been blasted by steam jets under break conditions in a BWR. The fragments obtained from these tests have then been used as initial specimens for further quasi-1D experiments using a water column test facility to study their settling properties, and to determine their drag coefficients.

In a separate test rig, the influence of debris-loaded strainers on pressure drop across them has also been investigated. Correlations from filter bed theory developed in other industries were adapted to fit the experimental findings, and used to model flow resistance as a function of particle load, filter bed porosity, and the parameters characterising the coolant flow. The aim was to derive formulae that may subsequently be used to model partially blocked strainers using CFD.

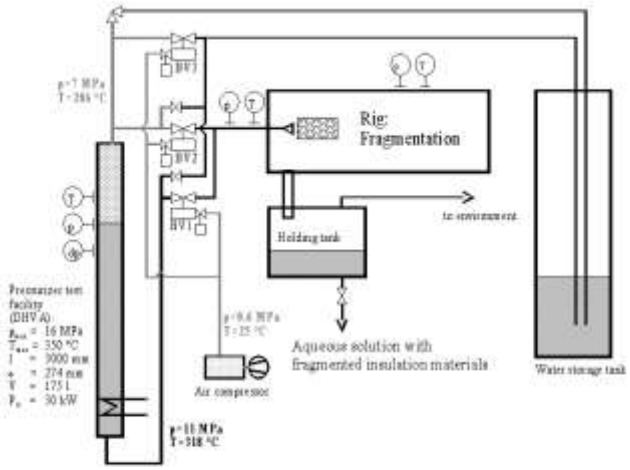


Fig. 1: Schematic of the fragmentation test rig.

The blast experiments carried out at the pressurizer test facility at Zittau/Görlitz is shown in schematic form in Fig. 1. The tests aim to quantify the fragmentation of different mineral wool insulation materials under typical LOCA conditions. The insulation material specimens (targets) were installed in the fragmentation vessel, and saturated steam up to 7 MPa (BWR-LOCA) pressure and saturated or subcooled water up to 11 MPa (PWR-LOCA) were applied. As a result of these experiments, fragmented insulation materials of the type seen in Fig. 2 were produced.

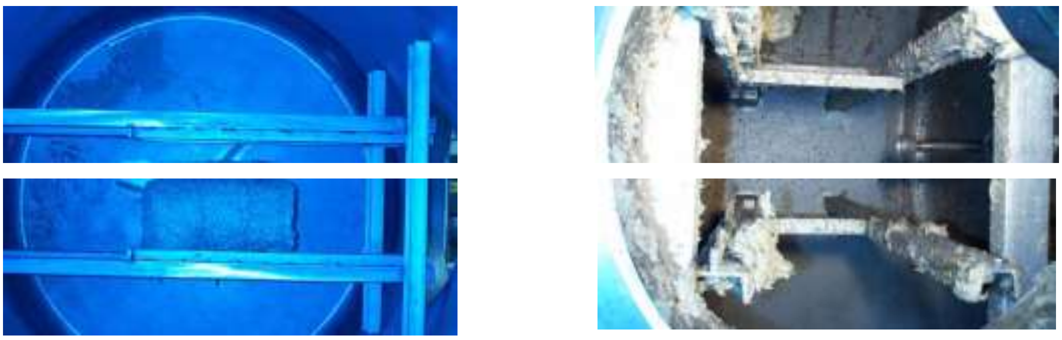


Fig. 2: Mineral wool specimen (left) and debris of fragments after a BWR-LOCA (right).

The settling behaviour of the insulation fragments in aqueous solution was studied in the test column shown in Fig. 3. The facility consists of a vertical, rectangular column made from acrylic glass. At the start of each test, the column is filled with water. It is possible to heat up the water up to 70°C by means of an external water circuit. The fragments were introduced at the top of the column and allowed to settle. The measurements taken during the settling process were:

• x-y paths of the settling fragments,
• settling velocities of the insulation fragments,
• geometric properties, grey value, volume and shape parameters of individual fragments,
• solid phase concentration.

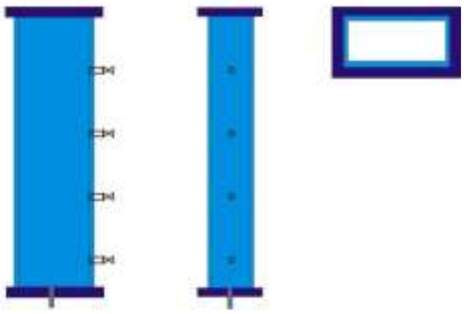



Fig. 3: Schematic and picture of the settling column test rig

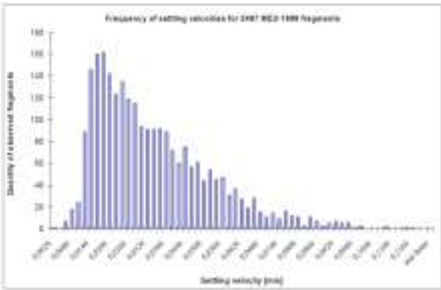


Fig. 4: Distribution of settling velocities for 2497 individual MD2-insulations fragments.

Digital image processing was applied for measuring insulation fragment geometries, their motions and velocities. A database of nearly 3000 fragments was compiled from the test data. The distribution of fragments as a function of the settling velocity is shown in Fig. 4. These data were used to derive appropriate drag coefficients for the accompanying CFD modelling.

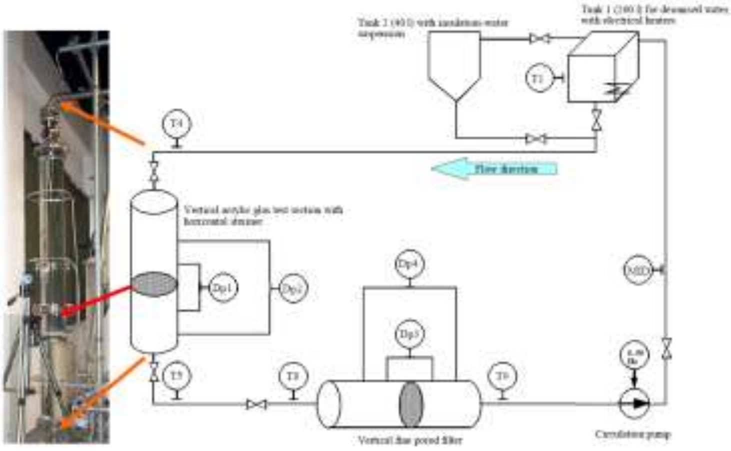
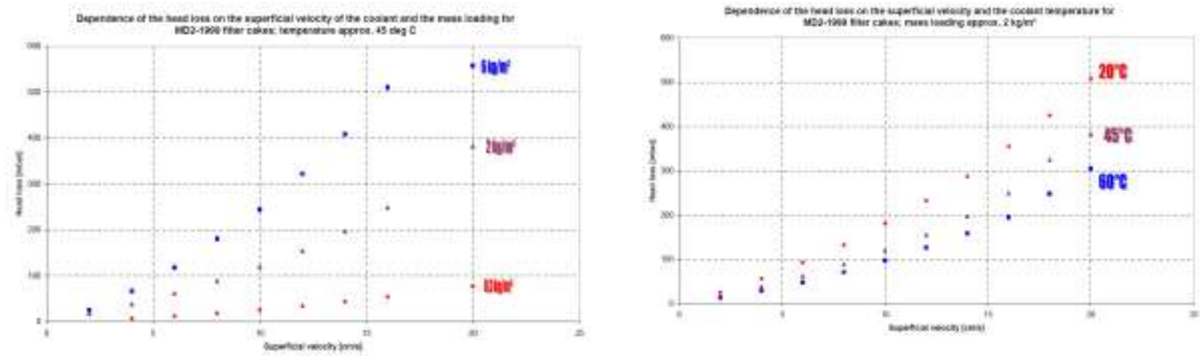


Fig. 5: Configuration of the test facility for measuring head loss across a model sump strainer in both vertical and horizontal positions.

In a separate test facility (Fig. 5), the pressure loss coefficient across a partially blocked sump screen was determined as a function of the mass loading of debris on the screen. The test facility consists of stainless steel components (storage tanks and pipes) and acrylic glass flow tracks, and can be operated in the temperature range 10°C to 70°C, under atmospheric pressure conditions. The insulation material under investigation (MD2-1999) was first fragmented at 7 MPa steam pressure using the fragmentation test rig (Fig. 1) under conditions appropriate for LOCA conditions in a BWR. The insulation material fragments (and the water carrier fluid) were then introduced into the holding tank without being previously dried. The measured head losses, as functions of mass loading and temperature are shown in Fig. 6.



(a) function of mass loading (b) function of temperature

Fig. 6. Head losses at horizontal MD2-1999 filters

With the information obtained from the separate-effects tests, a further series of experiments was performed to investigate particular the influence of particular geometric aspects on the sump clogging process. A schematic of the experimental set up is shown in Fig. 7. The water circulates in a race-track-type channel in the direction shown by the arrows, driven by the two impellers. Optionally, baffles are placed in the channel to investigate the influence on the deposition properties of the fibres induced by disturbances in the flow field.

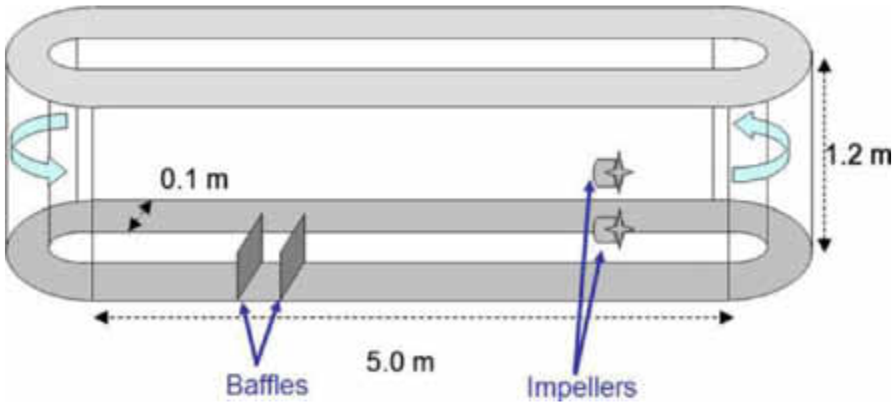


Fig. 7. “Racetrack” channel for the investigation of deposition and re-suspension of fibres.

The channel is of width 0.1 m, depth 1.2 m, and comprises two straight sections of length 5 m and bends with a radius of 0.5 m. The bulk water flow velocity can be varied between 0.01 m/s and 1.0 m/s. The fibre distribution and the water velocity field are observed using high-speed video and laser-based Particle Imaging Velocimetry (PIV) techniques. When in place, the baffle plates measure 0.1 m and 0.2 m in height, separated by a distance of 0.3 m.

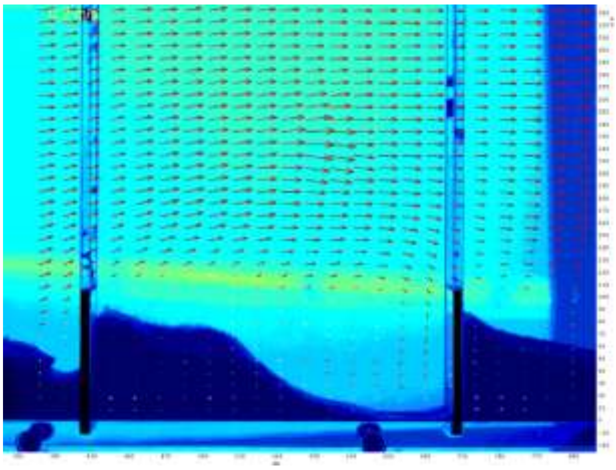


Fig. 8. Image obtained from PIV measurements of the velocity field and the fibre distribution between the baffles.

A typical vector velocity map obtained using PIV is reproduced in Fig. 8. The flow stream above the baffles remains largely undisturbed, except for the flow acceleration induced by the reduced channel flow area. Below the baffles, there is the expected break-up of the flow field, with a clearly recognisable recirculation region established between the baffles, and almost stagnant conditions upstream and downstream from this. The dark shaded areas show the regions of fibre deposition. As expected, this is enhanced in the low-flow regions.

From the outset, data from the experiments were intended to provide exactly defined flow boundary conditions for the accompanying CFD simulations. For the preliminary CFD investigations, the flow conditions were obtained for water flow in the channel in the absence of debris transport. The pumps were simulated as momentum sources, the source strength being adjusted to give the observed channel velocity. It could be seen from the calculations that the U-bends in the channel at the ends of the straight sections had a smoothing effect on the vertical flow profile. To provoke a flow disturbance, a model was developed to include the presence of the flow baffles, and simulation results compared directly against PIV data. The excellent agreement obtained for pure water flow conditions served as an essential starting point for the further investigations of fibre laden flow.

The principal challenges for the CFD modellers were to define suitable drag coefficients for the fibres, and to correctly account for their dispersion as a result of the turbulence in the water stream. Data obtained from the special-effect tests provided valuable information on these aspects. In particular, the settling velocities of the fibre material measured in the water column tests enabled appropriate drag coefficients to be derived, and other physical properties of the fibre phase, both necessary for the CFD model. The deposition and re-suspension behaviour of the fibres at low velocities was then investigated in the race-track channel geometry. From measurements taken during the pressure drop tests a CFD model, based on a porous medium approach with appropriate resistance factors, could be developed, and used to calculate the pressure drops across the strainers. Correlations were needed for the flow resistance caused by the fibre particle deposition. Initially, these were taken from the filter theory used in chemical engineering applications, but then adapted to the experiments. This approach also provided resistance coefficients for partially blocked strainers.

With all information in place, the sedimentation and re-suspension properties of the fibres observed in the race-track test could be examined, especially for the region between the baffles. As seen in Fig. 8, the presence of the baffles in the straight sections not only disturbs the motion of the carrier liquid (water), but also promotes deposition of the insulation debris. The experiments have revealed that the fibres agglomerate at a critical fibre volume fraction, which is manifested by a strong increase of the mixture viscosity. In addition, the fibres are deposited at the bottom of the channel below a critical water velocity of about 0.1 m/s, particularly at locations downstream of the obstacles. However, increasing the water velocity beyond 0.1 m/s causes the fibres to be re-mobilised, and become carried along with the prevailing flow stream.

The experiments carried out at HZDR, and the supporting analytical work performed by HZDR, have produced valuable data and numerical insights, respectively, into the effects of strainer clogging on decay heat removal following a LOCA incident. A broad database has been established from data produced from separate-effect tests for MD2-1999 mineral wool insulation material under settling, sedimentation, re-suspension and head loss build-up at horizontal strainers, has also been measures, all of which can be used for validating CFD models. The work was carried out under the terms of a joint collaboration agreement, but valuable data have been released in the open literature, and are available for CFD model development.

Ref. 1: Taylor, J.M. “Progress of resolution of generic safety issues”, US NRC Report SECY-96-092, May 1996.

Ref. 2: “Knowledge Base for the Effect of Debris on Pressurized Water Reactor Emergency Core Cooling Sump Performance”, NUREG/CR-6808, LA-UR-03-0880, Feb. 2003.

Ref. 3: NEA/NRC Workshop on Debris Impact on Emergency Coolant Recirculation, Albuquerque, NM, USA, Feb. 2004 (CD-ROM).

Ref. 4: Sandervåg, O. “Knowledge Base for Strainer Clogging - Modifications Performed in Different Countries since 1992”, OECD Nuclear Energy Agency report, NEA/CSNI/R(2002)6, Oct. 2002.

Ref. 5: Alt, S., et al. “Experiments for CFD Modelling of Cooling Water and Insulation Debris Two-Phase Flow Phenomena during Loff of Coolant Accidents”, Paper 22, NURETH-12, Pittsburg, PA, USA, Sept. 30 – Oct. 4, 2007 (CD-ROM).

Ref. 6: Grahn, A.; Krepper, E.; Alt, S.; Kästner, W. “Modelling of differential pressure buildup during flow through beds of fibrous materials”, *Chemical Engineering & Technology*, **29**(8), 997-1000 (2006).

Ref. 7: Grahn, A.; Krepper, E.; Alt, S.; Kästner, W. “Implementation of a strainer model for calculating the pressure drop across beds of compressible, fibrous materials”, *Nuclear Engineering and Design*, **238**, 2546-2553 (2008).

Ref. 8: Grahn, A.; Krepper, E.; Weiß, F.-P.; Alt, S.; Kästner, W.; Kratzsch, A.; Hampel, R. “Implementation of a pressure drop model for the CFD simulation of clogged containment sump strainers”, *Journal of Engineering for Gas Turbines and Power - Transactions of the ASME*, **132**, 082902 (2010).

Ref. 9: Höhne, T.; Grahn, A.; Kliem, S.; Weiss, F.-P, “CFD simulation of fibre material transport in a PWR under loss of coolant conditions”, *Kerntechnik*, **76**, 39-45 (2011).

Ref. 10: Krepper, E.; Cartland-Glover, G.; Grahn, A.; Weiss, F.-P.; Alt, S.; Hampel, R.; Kästner, W.; Seeliger, A., “Numerical and experimental investigations for insulation particle transport phenomena in water flow”, *Annals of Nuclear Energy*, **35**, 1564-1579 (2008).

Ref. 11: Krepper, E.; Weiß, F.-P.; Alt, S.; Kratzsch, A.; Renger, S.; Kästner, W. “Influence of air entrainment on the liquid flow field caused by a plunging jet and consequences for fibre deposition”, *Nuclear Engineering and Design*, **241**, 1047–1054 (2011).

6. IDENTIFICATION OF GAPS IN TECHNOLOGY AND ASSESSMENT BASES

As mentioned in the preceding section, an assessment matrix for a given application should comprise three groups of items:

- Verification problems with "highly-accurate" CFD solutions;
- Validation experiments and their CFD simulations;
- Demonstration simulations, possibly with some suitable supporting experiments.

Identification of gaps in the assessment matrices for a given application is possible only after thorough analysis of corresponding exact solutions and experiments, and their CFD counterparts. More than twenty NRS specific cases where CFD could bring substantial benefit were identified in Chapter 3. Analysis of such a large number of NRS problems to identify specific knowledge gaps represents an enormous task. Here, therefore, only some general guidance is given.

Verification Matrix

Code verification activities can be subdivided into Numerical Algorithm Verification, and Software Quality Assurance Practices. Here, only the Numerical Algorithm Verification will be discussed in which CFD solutions are compared with "correct answers", which are highly accurate solutions for a set of well-chosen test problems. Two pressing issues appear in designing and performing the Numerical Algorithm Verification:

- There is a hierarchy of confidence in these "highly accurate solutions", ranging from high confidence of exact analytical solutions and/or application of the Method of Manufactured Solutions (MMS), through semi-analytic benchmark solutions (reduction to numerical integration of ODEs) to highly accurate benchmark numerical solutions to PDEs.

- It is necessary to select application-relevant test problems, which in most industrial cases includes both complex physics and geometry.

Analytical solutions (closed solutions in the form of infinite series, complex integrals and asymptotic expansions to special cases of the PDEs that are represented in the conceptual model) are the basic and traditional tool of verification. Typically, inviscid or laminar flows in simple geometries can be treated analytically, so that only limited features of the CFD computer codes (or, more precisely, of the conceptual models) can be verified in this way.

One possible approach to expand the verification domain of CFD computer codes for problems with complicated physics (like turbulent flows) is represented by the *Method of Manufactured Solutions* (MMS). This method of custom-designing verification test problems proceeds roughly in the following steps:

- A specific form of the solution function is assumed to satisfy the PDE of interest.

- This function is inserted into the PDE, and all the derivatives are analytically derived.

- The equation is rearranged such that all remaining terms in excess of the terms in the original PDE are grouped into an algebraic forcing-function or source term on the right hand side of the equation.
- This source term is then simply added to the original PDE so that the assumed solution function satisfies the new PDE exactly.
- The boundary conditions of the Dirichlet, Neumann, or mixed type for the new PDE are calculated from the assumed solution function.
- The new PDE is then solved by the code to be verified and the result compared with the assumed solution function.

This method therefore requires that the computed source term(s) and boundary conditions are programmed into the code, which can represent a drawback. Not all CFD computer codes (mainly the commercial ones) provide such access to the source modules for those users developing, for example, their own physical models. Moreover, the difficulties associated with complex geometries are still present.

Application of numerical benchmarks requires thorough and well-documented verification of the code on simpler cases, very comprehensive numerical error estimation, and accurate calculations of the same case with independent experts, preferably using different numerical approaches and computer codes.

There is also a tendency to use some separate-effect experiments not only for development and validation of physical models, but also for conceptual model verification. Here, similar requirements to those related to numerical benchmarks must be met, not only by the computational solutions but also by the experiments. Only well designed, performed and documented experiments should be used. Such an activity represents in fact an interface between verification and validation on unit problems.

The primary responsibility for numerical algorithm verification should be placed upon the code developers, but code users should have access to the relevant, properly documented, information.

Validation and Demonstration Matrices

According to the tiered approach to validation of conceptual models, four progressively simpler levels of validation experiments,

- complete system,
- subsystem cases,
- benchmark cases, and
- unit problems

should be selected or proposed for each intended application of the CFD code, with at least one suitable experiment (or a set of experiments in the case of unit problems and benchmark cases) at each level.

Unit problems are characterized by very simple geometries and a limited number (preferably one) of important physical processes, since such experiments are very frequently aimed at development of physical models. Validation of a CFD conceptual model should start at this level. Repeated experimental runs are frequently possible, so that systematic errors can be detected. All the important code input data, initial conditions and boundary conditions can, in principle, be accurately measured. In some cases, and only at this level, multiple CFD computations are possible, enabling determination of probability of the output quantities. Possible gaps are represented by missing significant parameters, or measurement of such parameters at unsuitable locations, missing error analysis and, in the CFD simulations, missing analysis of possible effects of estimated values of quantities not measured in the experiment, on the computed results.

Benchmark cases typically involve only two or three types of coupled flow physics in more complex geometry than in the unit problems. Possible gaps at this level are in fact the same as in the case of unit problems, but they are more frequent. As to the CFD simulations, problems with demonstration of grid-independence of the solution are encountered.

Subsystem cases are at present the most complex cases solvable by a CFD code alone. It is difficult, and sometimes impossible, to quantify most of the test conditions required for CFD modelling, so estimation of the possible effects of such missing information on CFD simulation is essential. Computational grids are generally large, and grid independence cannot be proved in most cases. When meeting differences in measured and computed data, it is usually impossible to identify the cause of the differences, especially when CFD simulations at the unit and benchmark levels have not been performed. CFD simulations at the subsystem levels are very frequently close to demonstration simulations – it is sometimes difficult, if not impossible, to determine the degree to which the conceptual model simulates the reality.

As a complete system, the computational domain covered so far by system codes is understood here. At the complete system level, coupled CFD and system codes represent the only realistic approach. Verification and validation of such coupled codes is more complicated than verification and validation of either CFD or system code alone. The coupling itself can often be a source of errors. Validation of such coupled codes should be able to detect these errors if they are present. The unsteady nature of most problems met in nuclear reactor safety applications makes such identification even more difficult than for the steady problems. This field warrants more extensive research before application of such coupled codes becomes routine.

To summarize, validation of CFD codes for NRS application frequently encounters deficiencies, which includes (but is not restricted to):

- Phenomena Identification and Ranking Table (PIRT) for the intended application is not prepared.
- Quantified estimates of experimental and numerical uncertainties are not provided.
- Validation metrics, figures of merit or target values for the intended application are not clearly defined.
- Experiments, selected for validation at some of the tiers do not meet requirements put on validation experiments. Since validation experiments are very expensive, experiments intended for other purpose (e.g. for study of physical phenomena or for development of physical models), or very old experiments performed on already non-existing facilities (which excludes feedback between CFD simulations and experiments), are sometimes used.
- For some physical phenomena identified in the PIRT, suitable experiments are missing, so that new experiments must be proposed.
- Validation simulations cannot provide information on boundaries of regions of acceptability of the conceptual model from regions where the model cannot be applied, or where its application is questionable.

Demonstration simulations are frequently similar to subsystem or complete system cases when there is no or very limited experimental support. Only very approximate conclusions on applicability of the conceptual model can therefore be formulated. Nevertheless, demonstration simulations are very important from the viewpoint of application, since such simulations can support decisions on funding of verification and validation activities, or even of purchase of a CFD code. Especially at the complete system levels, multi-scale and multi-physics coupling is frequently required, and balance of resource constraints, including time, level of effort, available expertise and desired fidelity is very important. In many cases, a demonstration simulation is the first step in application of a CFD code to an NRS issue; such simulation can provide an insight into problems very probably encountered in future, more serious, application of the code. These problems can then be taken into account during planning of the code validation activity.

When demonstration simulations of the same problem are performed with two or more CFD codes, some idea on effectiveness of algorithms can be deduced. Since requirements put on the demonstration simulations are very relaxed in comparison with the validation simulations, it is not in fact possible to speak about "deficiencies", with the exception of formulation of the initial and boundary conditions (which are either deduced from system code calculations or defined as "the most unfavourable" from the point of view of the intended application), fineness of the computational grid, selection of time steps, and selection of physical models. An important role in the evaluation of demonstration simulations is played by expert judgement, which should take into account all the mentioned deficiencies.

Ref. 1: Mahaffy J. et al.: "Best Practice Guidelines for the use of CFD in Nuclear Reactor Safety Applications", NEA/CSNI/R(2007)5.

Ref. 2: Oberkampf W. L., Trucano, M.: "Design of and Comparison with Verification and Validation Benchmarks", Proc. Int. Workshop on Benchmarking of CFD Codes for Application to Nuclear Reactor Safety (CFD4NRS), Garching, Munich, Germany, 5-7 September 2006 (CD-ROM).

Ref. 3: Smith B. L. et al.: Assessment of Computational Fluid Dynamics (CFD) Codes for Nuclear Reactor Safety Problems, NEA/SEN/SIN/AMA(2005)3, OECD, May 2005).

6.1 Isolating the CFD Problem

Relevance of the phenomenon as far as NRS is concerned

Traditional 1-D system codes need to be "manipulated" to take account of 3-D effects, when the multi-dimensional aspect needs to be taken into account during the safety analysis. A local 3-D CFD computation is required in such cases to produce more trustworthy results.

What the issue is?

The issue arises of being able to isolate the 3-D analysis, where it is required, since in most situations there is a strong feed-back from the system parameters and it is presently inconceivable that CFD approaches will be able to be applied to the entire system.

What the difficulty is and why CFD is needed?

Flows in the upper and lower plena and downcomer of the RPV, and to some extent the core region, are all 3-D, particularly if driven by non-symmetric loop operation. Natural circulation and mixing in containment volumes are also 3-D phenomena. The number of meshes needed is far beyond the capabilities of present computers, closure relations for 3-D multi-phase situations are essentially non-existent, and criteria for defining flow regimes at the fine-mesh, CFD level is grossly underdeveloped, and no readily available CFD code has a neutronics modelling capability. With CFD not being mature enough to model the entire system, an alternative strategy is needed. Most attractive is to couple the existing 1-D system codes with the 3-D CFD codes in some way.

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, a problem remains in specifying the initial conditions (of velocities and field variables) for the CFD run 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 continuing the system analysis. 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.

What has been attempted and achieved/what needs to be done (recommendations)?

Several attempts have been made to couple CFD and system codes. Details are given in Section 6.9 of this document.

6.2 Range of Application of Turbulence Models

Relevance of the phenomenon as far as NRS is concerned

Almost exclusively, CFD simulations of NRS problems involve turbulent flow conditions.

What the issue is?

The turbulence community has assembled and classified a large selection of generic flow situations (jets, plumes, flows though tee-junctions, swirling flow, etc.), and made recommendations of which turbulence models are most appropriate. Care is needed to ensure that in NRS applications the turbulence model has been chosen appropriately.

What the difficulty is?

CFD is not capable of modelling entire reactor systems, which means that sections of the system must be isolated for CFD treatment. The range of scales can be large (e.g. in containments), and/or the flow phenomena rather special (e.g. ECC injection). It is necessary to extend the database of recognised flow configurations to include those particular to NRS applications of CFD, and build a suitable validation base.

What has been attempted and achieved/what needs to be done (recommendations)?

A very good exposé of this issue is given in the ECORA BPGs, so only a sketch will be given here.

In most industrial applications of CFD, RANS models are employed. However, due to the averaging procedure, information is lost, which has then to be fed back into the equations via an appropriate turbulence model. The lowest level of turbulence models offering sufficient generality and flexibility are two-equation models. They are based on the description of the dominant length and time scale by two independent variables. More complex models have been developed, and offer more general platforms for the inclusion of physical effects. The most complex are Second Moment Closure (SMC) models. Here, instead of two equations for the two main turbulent scales, the solution of seven transport equations for the independent Reynolds stresses and one length (or related) scale is required.

The challenge for the user of a CFD method is to select the optimal model for the application at hand from the models available in the CFD method. It is not trivial to provide general rules and recommendations for the selection and use of turbulence models for complex applications. Two equation models offer a good compromise between complexity, accuracy and robustness. The most popular models are the standard k-ε model and different versions of the k-ω model. However, the latter shows a severe free-stream dependency, and is therefore not recommended for general flow simulations, as the results are strongly dependent on user input.

An important weakness of standard two-equation models is that they are insensitive to streamline curvature and system rotation. Particularly for swirling flows, this can lead to an over-prediction of turbulent mixing and to a strong decay of the core vortex. There are curvature correction models available, but they have not been generally validated for complex flows. On the other hand, SMC models are much less robust, and it is often recommended to perform a first simulation based on the k-ε model, and use this as a starting point for the SMC approach. However, such an approach is hardly feasible for transient simulations, which are usually required for NRS applications.

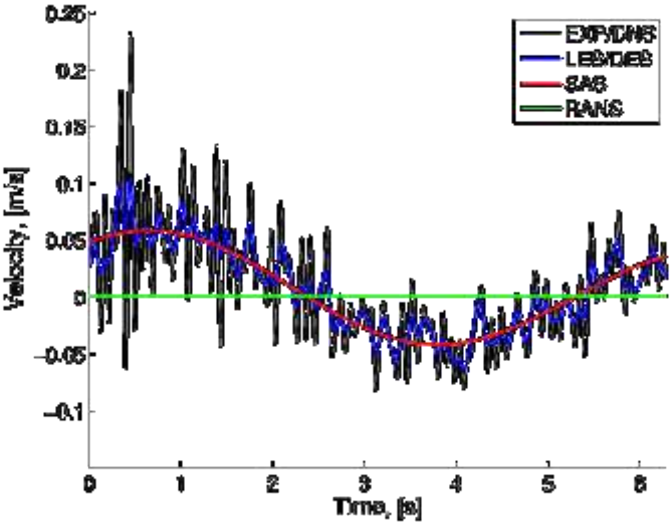
The first alternative to RANS is URANS (Unsteady RANS) or VLES (Very Large Eddy Simulation). The former is more descriptive of the actual technique of application: i.e. to carry out an unsteady RANS analysis, even if the boundary conditions are steady. Thus, if steady-state RANS calculation does not converge, it may be that some unsteady behaviour is present in the flow, such as periodic behaviour, plume or jet meandering, vortex shedding, etc. A URANS calculation can often identify the unsteady component, but it has to be remembered that averaging over all turbulence scales remains implicit in the method, and may not be appropriate to reliably capture the non-steady phenomena.

The amount of information to be provided by the turbulence model can be reduced if the large time and length scales of the turbulent motion are resolved explicitly. In LES, the equations are filtered over the grid size of the computational cells. All scales smaller than that provided by the resolution of the mesh are modelled using a suitable Subgrid Scale (SGS) model, and all scales larger than the cells are computed explicitly. Away from boundaries, LES appears trustworthy, even with very simplistic SGS models, such as Smagorinsky. In the wall regions, pure LES becomes very inefficient due to the need to scale the lateral dimensions in the same way as in the normal direction to capture the smaller scale eddies. This is not necessary in RANS, because the mean flow parallel to the wall changes much less abruptly than in the normal direction. Also, lack of sophistication of the SGS models may be tolerated in the bulk flow, but near walls the SGS stresses become much more important, and need to be accounted for accurately.

An alternative, is to entrust the entire boundary layer treatment to a RANS model for the “attached” eddies, and only use LES away from the walls, where the eddies are “detached”. This approach has become known as Detached Eddy Simulation (DES), and leads to considerable savings in CPU time. The case for continued use of LES in near-wall regions, probably in combination with a more complex SGS model, has to be judged in terms of possible information lost using DES versus the extra computational effort. This remains an active research area, particularly in the aerospace industry.

The Scale-Adaptive Simulation (SAS) model is a hybrid approach similar to DES, but operates without an explicit grid dependency. The controlling parameter is the ratio of the turbulent length scale L, for example, derived from the two-equation k-kL RANS model of Rotta (1972), and the von Karman length scale LvK, which is determined in the usual way from the first and second velocity gradients. In regions where the flow tends to be unstable, LvK is reduced, increasing the length scale ratio L/LvK. This leads to a reduction in the eddy viscosity. The flow will become more unstable, and hence transient in these regions, with vortices down to the scale of the local grid size being resolved, resulting in a LES-like behaviour. In stable flow regions, LvK remains large, which leads to high values for the eddy viscosity. In these areas, the model acts like a RANS model. Due to the model’s ability to resolve the turbulent spectrum, it is termed a “scale-adaptive simulation” model. It has similarities to the DES model, but has the advantage that it is not based on the local grid size and therefore avoids grid sensitivity problems.

As way of illustration, the picture shows how each approach to turbulence modelling is expected to capture an instantaneous velocity signal, produced experimentally or using Direct Numerical Simulation (DNS).



As a general observation, LES simulations do not easily lend themselves to the application of grid refinement studies, for either the time or space domains. The main reason is that the turbulence model adjusts itself to the resolution of the grid. Two simulations on different grids may not be compared by asymptotic expansion, as they are based on different levels of the eddy viscosity, and therefore on a different resolution of the turbulent scales. From a theoretical standpoint, the problem can be avoided if the LES model is not based on the grid spacing but on a pre-specified filter-width. This would allow grid-independent LES solutions to be obtained. However, LES remains a very expensive approach to turbulence modelling, and systematic grid and time step studies too prohibitive, even for a pre-specified filter. It is one of the disturbing facts that LES does not lend itself naturally to the application of BPGs.

Ref. 1: P. R. Spalart, "Strategies for turbulence modelling and simulations", Int. J. Heat and Fluid Flow, **21**, 252-263 (2000).

Ref. 2: Fureby, C., Tabor, G., Weller, H.G., Gosman, A.D., "A comparative study of subgrid scale models in homogeneous isotropic turbulence", *Phys. Fluids*, **9**(5), 1416 (1997).

Ref. 3: Menter, F. "CFD Best Practice Guidelines for CFD Code Validation for Reactor-Safety Applications", ECORA BPGs, 2002.

Ref. 4: Menter, F. and Y. Egorov: 2004, 'Revisiting the turbulent scale equation', in: *Proc.IUTAM Symposium in Goettingen; One hundred years of boundary layer research*.

Ref. 5: Menter, F., Y. Egorov, and D. Rusch: 2006, 'Steady and unsteady flow modelling using the k-$\sqrt{kL}$ model', Proc. *5th International Symposium on Turbulence, Heat and Mass Transfer*. Dubrovnik, Croatia.

6.3 Two-Phase Turbulence Models

This is a two-phase phenomenon, which is covered fully in the WG3 document.

Orientation

Turbulence modelling seems to be presently limited to extrapolations of the single phase k-epsilon models by adding interfacial production terms. The limits of such approaches have already been reached, and multi-scale approaches are necessary to take account of the different nature of the turbulence produced in wall shear layers, and the turbulence produced in bubble wakes. Certainly, more research effort is required in this area.

6.4 Two-Phase Closure Laws in 3-D

This is a two-phase phenomenon, which is covered fully in the WG3 document.

Orientation

Increasingly, the two-fluid (sometime three-fluid, to include a dispersed phase) model is being adopted for the multi-phase CFD simulations currently being carried out. In this approach, separate conservation equations are written for each phase. These equations require closure laws representing the exchange of mass, momentum and energy between the phases. Except for rather particular flow regimes (separated phases, dispersed second phase) genera-purpose expressions for such closure laws requires extensive further development.

6.5 Experimental Database for Two-Phase 3-D Closure Laws

This is a two-phase phenomenon, which is covered fully in the WG3 document.

6.6 Stratification and Buoyancy Effects

Relevance of the phenomenon as far as NRS is concerned

Buoyancy forces develop in the case of heterogeneous density distributions in the flow. 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 temperature (e.g., thermal stratification).

Other contributions to this report have underlined the possible occurrence of stratification and buoyancy forces. For single phase flows, one can recall stratified flow developing in the case of Pressurised Thermal Shock (see Section 5.2), hot leg heterogeneities (see Section 3.8), thermal shock (Section 3.12), induced break (Section 3.14), and for natural convection in many relevant safety situations for GFRs and LMFBRs in the context of PAHR (Post Accident Heat Removal); see specific Sections. For two-phase flow problems, the reader is referred to the WG3 document, NEA/CSNI/R(2007)15. Stratification may be one of the significant phenomena in the case of thermal shock, under some small-break LOCA conditions (see Section 3.22 on the AP600), and for water-hammer condensation. Stratification and buoyancy effects may lead to thermal fatigue, to modification of condensation rates, and to difficulties in predicting the associated mixing processes.

What the issue is?

Stratified flows and buoyancy-induced effects take place in many parts of the flow circuit: main vessel, lower and upper plena, pipes, and hot and cold legs. Most of the time, the phenomena are associated with unsteady 3D flow situations. The issue is to derive a modelling strategy able to handle all the situations of relevance to NRS.

What the difficulty is and why CFD is needed?

These complex phenomena are difficult to take into account using a system-code approach, and CFD is needed to better predict the time evolution of such flows, in particular the mixing rate between flows of different temperature (stratification may limit the action of turbulence, while buoyancy may in some cases promote mixing), and, in case of two phase flows, the behaviour of the different phases of the flow and the associated condensation rate.

For the case of single-phase flows, there remain difficulties and uncertainties concerning the modelling of turbulence for such situations. The standard k-epsilon model is known to poorly take into account mixing in strongly buoyant situations, and more complex closures (e.g., the Reynolds Stress Model) may be recommended for obtaining satisfactory results (Ref. 1). Unfortunately, the RSM model is much less robust that the k-epsilon model, and it may be difficult, or even impossible, to obtain converged solutions in complex geometries. Additionally, two further issues may be underlined: (i) the transitional state of such flows is difficult to handle in some situations, and (ii) the use of wall functions may lead to uncertainties if they are not designed for buoyant situations. (CFD two-phase flow issues are covered in the appropriate sections.)

What has been attempted and achieved/what needs to be done (recommendations)?

Numerous CFD simulations have already been undertaken for specific situations, including the use of turbulence modelling, wall functions, etc. Due to the large number of the situations analysed, the main recommendation may concern the development of specific experiments to assess the validity range of the existing modelling capability.

Ref. 1: M. Casey, T. Wintergerste (Eds.), "ERCOFTAC Special Interest Group on Quality and Trust in Industrial CFD: Best Practice Guidelines", Version 1.0, January 2000.

6.7 Coupling of CFD code with Neutronics Codes

Relevance of the phenomenon as far as NRS is concerned

Precise prediction of the thermal loads to fuel rods, and of the main core behaviour, result from a balance between the thermal hydraulics and the neutronics.

What the issue is?

Basic understanding consists of recognising that the thermal hydraulics is coupled with the neutronics through the heat release due to neutronic activity (nuclear power distribution and evolution), and that 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. boron, see Section 3.7).

What the difficulty is and why CFD is needed?

The difficulty is to perform a coupled simulation, involving a CFD code adapted to the core description and a neutronics code, and to ensure consistent space and time precision of the two aspects.

What has been attempted and achieved/what needs to be done (recommendations)?

Some progress has been made in this area.

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, for both steady-state and transient situations (c.f. OECD/NEA benchmarks). Pin or cell level coupling has also been investigated.

The coupling between a CFD code (Trio_U) and a Monte-Carlo neutronics code (MCNP) has been tested in the context of a PhD programme for the MSRE prototype. The results obtained so far compare well with the experimental data. Their extrapolation suggests ways of improving the safety coefficients of power molten-salt reactors (Ref. 1).

CFD neutronic coupling between STAR-CD and VSOP is proposed in the case of PBMR (see Ref. 2).

Coupling between core thermal hydraulics and neutronics with the SAPHYR system [Ref. 3] is based on the FLICA4 3D two-phase flow model and the CRONOS2 3D diffusion and transport models.

Several benchmarks have been computed in the frame of OECD/NEA [Ref. 4]: PWR Main Steam Line Break [Ref. 5], BWR Turbine Trip [Ref. 6], and currently the VVER-1000 Coolant Transient (for which fine-mesh CFD models are used). CRONOS2 and FLICA4 have also been successfully applied to the TMI Reactivity Insertion Accident benchmark (with BNL and KI, Refs 7-8], with pin-by-pin modelling, and within the NACUSP project (5th European FP, Ref. 9].

The 3D model of FLICA4 takes into account cross-flows between assemblies, related to core inlet boundary conditions or neutronic power distribution. Feedback parameters, such as fuel temperature and moderator density, are computed at the fuel assembly level, without collapsing several assemblies into macro-channels, which results in a better accuracy for local parameters of interest for safety: i.e. power peak and maximum fuel temperature. For conditions in which there is large asymmetry, like rod ejection or main steam-line break(SLB), FLICA4 features a two-level approach (zoom): the assembly level and the sub-channel level, either by coupling two FLICA4 calculations (exchange of boundary conditions), or by using a non-conforming mesh.

The coupling of another CFD code (CAST3M) with the neutronics code (CRONOS2) has been performed by CEA for the core of a gas-cooled reactor (GTMHR), in order to evaluate feedbacks (Ref.1 11). Similar work is being performed at Framatome, with the development of the coupling of the STAR-CD code with the CRONOS2 code.

Possible improvements would be (i) the coupling of CFD codes with more advanced (i.e. deterministic or stochastic transport) neutronics models; (ii) the development of a multi-scale approach, in order to optimise the level of description with the conditions, since, in many 3D cases, the power is very peaked (rod ejection, boron dilution, SLB, etc.), and fine-scale models could be used only in a limited region; and (iii) the development of time-step management procedures for complex transients in which the thermal hydraulics and neutronics time-scales are not the same.

Ref. 1: F. Perdu “Contributions aux études de sûreté pour des filières innovantes de réacteurs nucléaires”, PhD thesis, Université Joseph Fourier Grenoble, 2003.

Ref. 2: http://www.cd-adapco.com/news/18/reactor.htm.

Ref. 3: C. Fedon-Magnaud et al. “SAPHYR: a code system from reactor design to reference calculations”, M&C 2003 (ANS), Gattlinburg, Tennessee, April 6-11, 2003.

Ref. 4: http://www.nea.fr/html/science/egrsltb.

Ref. 5: Caruso, A., Martino, E., Bellet, S., "Thermal-hydraulic behavior inside the upper upper plenum and the hot legs of A 1300 MW PWR: Qualification on BANQUISE mock-up and application to real reactor", American Society of Mechanical Engineers, Pressure Vessels and Piping Division (Publication) PVP, 431, pp. 155-162, 2001

Ref. 6: Caruso, A., Martino, E., Bellet, S., "3D numerical simulations of the thermal-hydraulic behavior into the upper plenum and the hot legs of a 1300 MW PWR configuration : Qualification on BANQUISE mock-up", American Society of Mechanical Engineers, Pressure Vessels and Piping Division (Publication) PVP, 414, pp. 117-121, 2000

Ref. 7: P. Ferraresi, S. Aniel, E. Royer, “Calculation of a reactivity initiated accident with a 3D cell-by-cell method: application of the SAPHYR system to the TMI1-REA benchmark”, CSNI Workshop, Barcelona, April 2000.

Ref. 8: J.C. Le Pallec, E. Studer, E. Royer, “PWR Rod Ejection Accident: Uncertainty analysis on a high burn-up core configuration”, Int. Conf. On Supercomputing in Nuclear Applications (SNA). Paris, 2003.

Ref. 9: K. Ketelaar et al. « Natural Circulation and Stability Performance of BWRs (NACUSP)”, FISA-2003, Luxembourg, November 10-13, 2003.

Ref. 10: E. Studer et al., “Gas-Cooled Reactor Thermal-Hydraulics using CAST3M and CRONOS2 codes”, Proc. 10th Int. Topical Meeting on Nuclear Thermal-Hydraulics, NURETH-10, Seoul, Korea, October 5-9, 2003.

Ref**. 11**: Höhne, 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, Nuclear Engineering and Design, 236(12), 1309-1325 (2006)

Ref. 12: Höhne, T.; Kliem, S.; Rohde, U.; Weiss, F.-P., Buoyancy driven coolant mixing studies of natural circulation flows at the ROCOM test facility using ANSYS CFX, 14th International Conference on Nuclear Engineering, ASME, 16-20 July, 2006, Miami, USA CD-ROM, Paper ICONE 14-89120.

6.8 Coupling of CFD code with Structure Codes

Relevance of the phenomenon as far as NRS is concerned

The 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 have consequences regarding plant safety. In the case of thermal-hydraulic issues relating to the containment, there are instances of chugging and flow-induced condensation producing jets in suppression pools in BWRs, and in large water pools for some evolutionary reactions in which the mechanical loads on submerged surfaces need to determined and the heat transfer to the walls have to be simulated simultaneously, usually by coupling implicitly a CFD code and structure code.

What the issue is?

In order to obtain detailed information on the thermal and/or pressure loads to the structures, CFD analysis of the flow field is often necessary. To facilitate the transfer of the load information, it is often desirable, and sometimes necessary, to directly link CFD and structure codes. If there is no feed-back of structural displacement on the flow field, it is sufficient to have a one-way coupling only, and the structural analysis can be performed “off-line” to the CFD simulation. However, if there is a feed-back, for example due to changes in flow geometry, a two-way coupling between the codes is needed, and the CFD and structural analysis must be computed simultaneously (or perhaps just iteratively in simple cases).

What the difficulty is and why CFD is needed?

The pressure loading to structures may be computed at different levels of sophistication. In simple cases, a static loading, estimated using lumped-parameter methods, may be input as a boundary condition to the stress analysis program. Similarly with thermal loading, provided a reliable estimate of the appropriate heat transfer coefficients are known. In these circumstances, the stress analysis may be performed independently of any associated CFD. However, if there are significant spatial variations in the loadings, it may be necessary to provide cell-by-cell information of the flow details. CFD is needed for this.

What has been attempted and achieved/what needs to be done (recommendations)?

The code coupling of the structural mechanics code ANSYS and the CFD code ANSYS-CFX has been applied for different aerodynamic test cases (Ref. 1). The analysis of a pitching airfoil demonstrates the performance of ANSYS-CFX for the prediction of the transient lift and momentum coefficients. Furthermore, the mechanical coupling example of an elastic-walled tube shows the flexible coupling concept between structural and fluid software. The combination of both, transient and flexible coupling is applied for the AGARD 445.6 wing flutter test. A good agreement has been obtained for the comparison of the flutter frequency in a wide range of Mach numbers. The technology for NRS-related issues, e.g. flow-induced vibrations, water-hammer, etc., would follow similar lines.

Coupling between STAR-CD and Permas is described on the Adapco website. The deformations and stresses of the Sulzer Mixer, subjected to high-pressure load, was investigated by coupling STAR-CD and Permas using MpCCI. The geometry model takes into account all the details of the structure, even welding points. The mixer structure was built entirely as a 3D solid model using Unigraphics. As a first step, the steady-state fluid flow was computed by STAR-CD without any code coupling. As a second step, the fluid forces were transferred from the fluid code to the stress code by coupling the codes. This method (one-way-coupling) assumes that the fluid flow topology is not affected by the structural displacement. This is realistic for the kind of mixer under consideration, and would be true also for many NRS applications involving heavy reactor components. The deformations, stresses and rotational movement agreed with experimental observations. Work on the full coupling of the flow and stress computations, requiring STAR-CD’s moving-mesh capability, is in progress. The use of STAR-CD, Permas and MpCCI provides more realistic computation of the forces on the structures, and better design and optimisation of the mixer geometry.

A very interesting approach to problems of fluid-structure interaction from the point of view of methodology is described in De Sampaio et al. (2002). The authors combine a remeshing scheme with a local time-stepping algorithm for transient problems. Since the solution at different locations is then not synchronized, a time-interpolation procedure is used to synchronize the computation. Turbulence is modelled via Large Eddy Simulation without an explicit sub-grid model; the effect of the unresolved sub-grid scales on the mean flow is performed by the numerical method used. This approach is called ‘implicit sub-grid modelling’ or ‘ILES’, and corresponds to ‘numerical LES’, see Pope (2004). The problem domain is split into an ‘external Eulerian region’, for the fluid far from the structure, a ‘transition region’, where an ALE reference frame is used, and a ‘Lagrangian description’ at the fluid-solid interface. The approach is validated on the problem of vortex shedding on a square cylinder.

Sauvage and Grosjean (1998) at ENSIETA in France have validated an iterative approach to modelling fluid-structure interaction. Their study examines the deformation of a thin aluminium slab in a cross-flow of air by coupling an FLUENT simulation of the airflow to an ABAQUS prediction of the structural deformation. Starting with a prediction of air flow around the non-deformed slab, the researchers determined the pressure forces on the slab, and used these as input to ABAQUS. The ABAQUS calculations predicted the slab deformation, which was used to redefine the FLUENT mesh defining the flow geometry. Using the modified mesh, the FLUENT calculations predicted new pressure forces as modified inputs to the ABAQUS run. By iterating between the two codes, convergence to a steady-state prediction of the flow around the deformed slab could be obtained. The calculation procedure was validated against wind tunnel test data on deformation and drag. Calculations were within about 3% of measurements for both quantities. Again, this technique has potential application to many NRS issues involving fluid-structure interaction.

CEA has made a study of the mechanisms leading to cracking in mixing zones of piping networks, as a result of thermal loading. The overall analysis was performed with a single computer code: the CAST3M code developed by CEA. Cracks appearing in a mixing tee, and its connection with the pipework in the Civaux Unit 1 were adequately explained by the various calculations made.

A run-time coupling using PVM (Parallel Virtual Machine) has been established between the codes COCOSYS (a lumped-parameter containment code) and ANSYS-CFX. The aim of the work was to replace certain user-specified locations of the domain described by COCOSYS by a ANSYS-CFX model, and to exchange the boundary fluxes of mass and energy between the codes on-line.

A comprehensive overview of experimental and theoretical work on flow-induced vibration of single and multiple tubes in cross-flow is described in Blevins (1990). In Kuehlert et al. (2006), the FLUENT 6.3 code with a simple two degrees of freedom spring and damper model was applied to study flow-induced vibration of individual tubes. The realizable k-epsilon model of turbulence in 2D was used at Re=3800. Good correspondence was found. For Re=3106 and a single tube, a demonstration analysis was made in 3D using the DES turbulence modeling approach. Validation of flow past stationary tube banks was made in preparation for a demonstration of tube oscillation. The FLUENT 6.3 code was coupled with the ABAQUS structural analysis code for this purpose, and the experimental data of Simonin and Barcouda (1988) were used. Both LES and RNG k-epsilon models of turbulence were tested in 3D.

Ref. 1: Kuntz, M., Menter, F.R., “Simulation of Fluid Structure Interaction in Aeronautical Applications”, to be published in the ECCOMAS 2004 Conference, July 2004.

Ref. 2: http://www.cd-adapco.com/news/16/fsiinnotec.htm

Ref. 3: Sauvage, S., Grosjean, F., "ABAQUS Married with Fluent," ABAQUS Users' Conference, Newport, Rhode Island, May 1998, pp. 597 – 602.

Ref. 4: Blevins R. D. *Flow-induced Vibration*, Van Nostrand Reinhold, New York 1990.

Ref. 5: De Sampaio P. A. B., Hallak P. H., Coutinho A. L. G. A., Pfeil M. S., “Simulation of turbulent fluid-structure interaction using Large Eddy Simulation (LES), Arbitrary Lagrangian-Eulerian (ALE) co-ordinates and adaptive time-space refinement”, Use of Computational Fluid Dynamics (CFD) Codes for Safety Analysis of Reactor Systems, Including Containment. Pisa, Italy, 11-14 November 2002.

Ref. 6: Hover F. S., Techet A. H., Triantafyllou, M.S. “Forces on oscillating uniform and tapered cylinders in cross flow”, *J. Fluid Mech.*, **363**, 97-114 (1998).

Ref. 7: Kuehlert K., Webb S., Joshl M., Schowalter D., “Fluid-structure interaction of a steam generator tube in a cross-flow using large-eddy simulation”, Proc. ICONE 14, July 17-20, 2006, Miami, USA.

Ref. 8: Pope S. B. “Ten questions concerning the large-eddy simulations of turbulent flows”, *New Journal of Physics*, **6**, 35 (2004).

Ref. 9: Simonin O., Barcouda M., “Measurements and prediction of turbulent flow entering a staggered tube bundle”, 4th Int. Symp. Of Applications of Laser Anemometry to Fluid Mechanics, Lisbon, Portugal, 1988.

6.9 Coupling CFD with System Codes: Porous Medium Approach

Validation of CFD-type computer codes on separate-effect experiments is discussed thoroughly in this document and in the companion Best Practice Guidelines (NEA/CSNI/R(2007)5). The process of validation in the context of nuclear reactor simulations are, in majority of cases, beyond the possibilities of present hardware if a CFD code is used alone. Use of a less detailed, less demanding system analysis code to produce initial and boundary conditions for the CFD code is a practical alternative. Such multi-scale coupling is indispensable in the case of demonstration simulations and, of course, application of a CFD code to real industrial problems. Moreover, in such problems it is very frequently necessary to simulate not only thermal-hydraulics, but also phenomena belonging to different fields of physics or even to chemistry. However, in this type of multi-physics coupling, problems with different spatial and temporal scales appear.

General methods of coupling are treated in several books and papers, e.g., Zienkiewicz (1984), Hackbush, Wittum (1995), Cadinu et al. (2007) and E et al. (2003). Most generally, couplings are distinguished between those taking place on the same domain, by changing the differential equations describing the corresponding physical phenomena (this approach is frequently realized by means of a single computer code), or coupling on adjacent domains by matching boundary conditions at thir interfaces. In this case, either the models are combined to produce a comprehensive model for the coupled problem (joint, or *simultaneous solution strategy*), or there are modules solving the individual problems, and coupling is effected via an outer iteration (changing of parameters, boundary conditions, or geometries after each step or selected steps of the outer iteration – *partitioned solution strategy*). Whenever an outer iteration is used, the problem of the optimum level of explicitness of the coupling has to be faced, especially when two-way coupling is required. Generally, explicit coupling is easy to program compared with implicit coupling, but is more prone to numerical instabilities.

Independently of the details of the particular coupling strategy, validation and assessment of the coupled code is required. The individual codes usually solve problems with different spatial and time scales and, particularly if two-way coupling is required, it is not enough to validate or assess the codes individually. Design of corresponding experiments must take into account different requirements concerning density of instrumentation (when multi-scale coupling of codes is tested) or requirements of different type of instrumentation (in the case of multi-physics coupling).

There are several examples of coupled CFD or CFD-type codes with system codes, as can be seen in the following Table, reproduced from Cadinu et al. (2007):

Table1: Examples of Coupled Codes

Authors, source	**System code**	**CFD code**	**Process**
Jeong et al. (1997)	RELAP5	COBRA/TF	LOFT L2-3 LOCA Experiment
Graf (1998)	ATHLET	FLUBOX	UPTF Experiment, Weiss et al. (1986)
Kliem et al. (1999)	ATHLET	CFX	MSLB analysis
Aumiller et al. (2002)	RELAP5	CFDS-FLOW3D	Subcooled boiling experiments Christensen (1961)
Gibeling, Mahaffy (2002)	Authors' 1D code	NPHASE	Pipe flow experiments Laufer (1953)
Schultz, Weaver (2003)	RELAP5	FLUENT	
Grgic et al. (2002)	RELAP5	GOTHIC	IRIS reactor 4-inch break

Coupling of the CAST3M/ARCTURUS CFD code with neutronics code CRONOS2 is described in Studer et al. (2005). The architectures of the coupling algorithm and sensitivity studies are described. The coupled code is aimed at applications to gas-cooled reactors. No validation has been possible so far, since experimental data including both thermal hydraulic and neutronic parameters are missing. The facility SIRIUS-F, built in Japan (see Furuya et al., 2007), could provide data for filling this gap.

An example of extensive research in the field of code coupling is the development of the methodology for coupling of the RELAP5 and RELAP5-3D codes to different codes, as described in Weaver et al. (2002), Schultz, Weaver (2002, 2003), Schultz et al. (2002), and Grgic et al. (2002). The coupling is performed via an *Executive Program*, originally based on a generic explicit coupling methodology, described in Aumiller et al. (2001) for coupling the CFX code with RELAP5-3D, and now also using semi-implicit coupling methodology, as described in Weaver et al. (2002). The RELAP5 code can be either master or slave process of the coupled codes. In the case of the coupling of RELAP5 and the CFD code FLUENT, the Executive Program monitors the calculational progression in each code, determines when both codes have converged, governs the information interchanges between the codes, and issues instructions to allow each code to progress to the next time step. The first round validation matrix for the RELAP5-3D/FLUENT coupled code, reproduced from Schultz et al. (2002), is shown in the Table below (the coupled code was intended for simulation of phenomena taking place during normal and transient operation of the pebble-bed modular reactor and other high-temperature gas reactor systems):

Table 2: Validation matrix for the FLUENT/RELAP5-3D coupled code

Case No.	Description	Working Fluid	Phenomena or Objective	Gas reactor Region of Interest	Reference
1	Turbulent flow in pipe section	Air	Mesh coupling between FLUENT & RELAP5-3D	Inlet pipe	Streeter (1961)
2	Turbulent flow in backward facing step with heat transfer	Air	1.Mesh coupling between FLUENT & RELAP5-3D 2.Flow profile	Inlet pipe and inlet plenum	Baughn et al. (1984)
3	Neutronic-fluid interaction in core region	Water	RELAP5/ATHENA neutronics coupling with FLUENT mesh.	Core (although this data set is for geometry unlike gas reactors, no data is available for gas reactors).	Ivanov et al. (1999)
4	Counter-current two-phase flow	Water & SF6	1.Mesh coupling between FLUENT & RELAP5-3D 2.Flow behaviour calculated by FLUENT	Potential pipe break and counter-current flow at break when unchoked.	Stewart et al. (1992)
5	Flow through packed-bed	Air	FLUENT's capability of calculating flow through portion of packed bed.	Core	Calis et al. (2001)
6	Air ingress	Helium & air	Evaluate coupled code's capability to calculate counter-current multi-species flow.	Primary pipe break	Hishida et al. (1993)

One of the problems of multi-scale coupling, i.e. the transition between 1D and 3D description at the interface, which is the case No. 1 of the RELAP5-3D and FLUENT validation matrix, was also studied by Gibeling & Mahaffy (2002). Application of uniform profiles for transmitted quantities at the interface is a common practice, even if using a stand-alone CFD code. The paper shows that this approach leads to erroneous pressure and temperature fields (fictitious entrance region).

The importance of consistent equations of state (EOS) in the coupled codes is stressed by Ambroso et al. (2005). The paper deals among other things with a 1D flow region separated into two sub-regions, both described by single set of equations, but with slightly different EOSs. In this situation, the saturated fluid leaving one solution domain may appear in the other solution domain as either sub-cooled or superheated fluid having a different temperature in the receiving domain from its temperature in the sending domain – see also Weaver et al. (2002). A similar statement is made by Schultz et al. (2002).

Clearly, a start has been made in the validation of CFD codes coupled to system (and neutronics) codes for NRS applications. It is anticipated that coupled codes will be used much more frequently in the future, and validation will remain a key issue. It is worth remarking again that it is necessary to perform verification and validation exercises for the component parts of a coupled code, but this is not sufficient to claim V&V for the coupled code itself: an additional programme is needed for this.

Ref. 1: Ambroso A. et al., Coupling of multiphase flow models. 11th International Topical Meeting on Nuclear Thermal-Hydraulics (NURETH-11), Avignon, France, October 2-6, 2005. Paper 184.

Ref. 2: Aumiller D. L., Tomlinson E. T., Bauer R. C.: A coupled RELAP5-3D/CFD methodology with a proof-of-principle calculation, *Nucl. Eng. Design,* **205,** 83-90 (2001).

Ref. 3: 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. Design*, **216**, 77-87 (2002).

Ref. 4: Cadinu F., Kozlowski T., Dinh T.-N.: Relating system-to-CFD coupled code analyses to theoretical framework of a multiscale method. Proc. ICAPP 2007, Nice, France, May 13-18, 2007. Paper 7539.

Ref. 5: Calis H. P. A., Nijenhuis J., Paikert B. C., Dautzenberg F. M., van den Bleek C. M.: CFD Modeling and Experimental Validation of Pressure Drop and Flow Profile in a Novel Structured Catalytic Reactor Packing, *Chemical Eng. Science*, **56**, 1713-1720 (2001).

Ref. 6: Chudanov v. v., Aksenova A. E., Pervichko V. A.: CFD to modeling molten core behavior simultaneously with chemical phenomena. The 11th Int. Topical Meeting on Nuclear Thermal-Hydraulics (NURETH-11), Avignon, France, October 2-6, 2005. Paper 048.

Ref. 7: E W., Engquist B., Huang Z.: Heterogeneous multiscale method: A general methodology for multiscale modelling, *Phys. Rev. B*, **67**, 92-101 (2003).

Ref. 8: Furuya M., Fukahori T., Mizokami S., Development of BWR regional stability experimental facility SIRIUS-F, which simulates thermal hydraulics-neutronics coupling, and stability evaluation of ABWRs, *Nucl. Technol.*, **158**, 191-207 (2007).

Ref. 9: Gibeling H., Mahaffy J.: Benchmarking simulations with CFD to 1-D coupling. Technical Meeting on Use of Computational Fluid Dynamics (CFD) Codes for Safety Analysis of Reactor Systems, Including Containment. Pisa, 11-14 November 2002.

Ref. 10: 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).

Ref. 11: Grgic D., Bajs T., Oriani L., Conway L. E.: Coupled RELAP5/GOTHIC model for accident analysis of the IRIS reactor. Technical Meeting on Use of Computational Fluid Dynamics (CFD) Codes for Safety Analysis of Reactor Systems, Including Containment. Pisa, 11-14 November 2002.

Ref. 12: Hackbusch W., Wittum G. (eds.): Numerical Treatment of Coupled Problems, Vol. 51 of *Notes on Numerical Fluid Mechanics*, Vieweg, 1995.

Ref. 13: Hishida M., Fumizawa M., Takeda T., Ogawa M., Takenaka S., Researches on air ingress accidents of the HTTR, *Nucl. Eng. Design*, **144**, 317-325 (1993).

Ref. 14: Ivanov K. N., Beam T. M., Baratta A. J.: PWR Main Steam Line Break (MSLB) Benchmark, Volume I: Final Specifications. NEA/NSC/DOC(99)8, April 1999.

Ref. 15: Jeong J. J., Kim S. K., Ban C. H., Park C. E.: Assessment of the COBRA/RELAP5 Code Using the LOFT l2-3 Large-Break Loss-of-Coolant Experiment. Ann. Nucl. Energy 24 (1997) 1171-1182.

Ref. 16: 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. Topical Meeting on Nuclear Reactor Thermal Hydraulics NURETH-9, San Francisco, California, October 3-8, 1999.

Ref. 17: Laufer J.: The structure of turbulence in fully developed pipe flow. NACA Report NACA-TN-2954, 1953.

Ref. 18: Schultz R., Wieselquist W.: Validation & Verification: Fluent/RELAP5-3D Coupled Code. 2001 RELAP5 User's Seminar Sun Valley, ID, September 2001.

Ref. 19: Schultz R. R., Weaver W. L.: Coupling the RELAP-3D© systems analysis code with commercial and advanced CFD software. Technical Meeting on Use of Computational Fluid Dynamics (CFD) Codes for Safety Analysis of Reactor Systems, Including Containment. Pisa, 11-14 November 2002.

Ref. 20: Schultz R., Weaver W. L.: Using the RELAP5-3D Advanced Systes Code with Commercial and Advanced CFD Software. Proc. 11th Int. Conf. On Nuclear Engineering, Tokyo, Japan. April 20-23, 2003.

Ref. 21: Schultz R. R., Weaver W. L., Ougouag A. M.: Validating & verifying a new thermal-hydraulic analysis tool. Proc. ICONE10, 10th Int. Conf. on Nuclear Engineering, Arlington, VA, April 14-18, 2002.

Ref. 22: Stewart W. A., Pieczynski A. T., Srinivas V.: Natural circulation experiments for PWR High Pressure Accidents. EPRI Project RP2177-5, 1992.

Ref. 23: Streeter V. L.: Fluids Handbook. McGraw-Hill, 1961.

Ref. 24: Studer E., Beccatini A., Gounand S., Dabbene F., Magnaud J. P., Paillere H., Limaiem I., Damian F., Golfier H., Bassi C., Garnier J. C.: CAST3M/ARCTURUS: A coupled heat transfer CFD code for thermal-hydraulic analyzes of gas cooled reactors. The 11th Int. Topical Meeting on Nuclear Thermal-Hydraulics (NURETH-11), Avignon, France, October 2-6, 2005. Paper 318.

Ref. 25: Weiss P., Sawitzki M., Winkler F.: UPTF, a Full Scale PWR Loss-of-Coolant Accident Experimental Program. Kerntechnik 49 (1986)

Ref. 26: Wieselquist W. A.: One validation case of the CFD software FLUENT: Part of the development effort of a new reactor analysis tool. Proc. ICONE10, 10th Int. Conf. on Nuclear Engineering, Arlington, VA, April 14-18, 2002.

Ref. 27: Weaver W. L., Tomlinson E. T., Aumiller D. L.: A generic semi-implicit coupling methodology for use in RELAP5-3D. Nucl. Eng. Des., 211, 13-26 (2002).

Ref. 28: Yamaguchi a., Takata T., Okano Y.: Multi-level modeling in CFD coupled with sodium combustion and aerosol dynamics in liquid metal reactor. Pisa 2002.

Ref. 29: Zienkiewicz O. C.: Coupled problems and their numerical solution. In Lewis R. W., Bettes P., Hinton E. (eds.): Numerical Methods in Coupled Systems, John Wiley & Sons, 1984.

6.10 Computing Power Limitations

The original version of *Parkinson's Law* (Ref. 1), "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 law's development included noting that as Britain's overseas empire declined in importance, the number of employees at the Colonial Office increased. 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”.

Despite the overwhelming amount of possibilities and advantages of present CFD codes, their role should not be exaggerated. 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, these system codes are recognised internationally. The physical models are based on reasonable assumptions concerning the steam and water flows, and their interaction. The circuits are treated as an assembly of 1D pipe elements, 0D volumes, and eventually some 3D component modelling. Intensive experimental programs of validation on system loops, or local component mock-ups, were carried out. So there is some confidence in their results, provided they are used in their domain of validation, and by experienced users.

Today, a large part of the system calculations are made on workstations or PCs. In the mid-term, say 5 to 10 years, it is foreseen to improve the two-fluid models, perhaps with extension to three fields to include droplets and bubbles, and incorporation of transport equations for interfacial area; 3D modelling would be used, as required. During the same period, the increasing computer efficiency will allow the use of refined nodalisation, and the capture of smaller scale phenomena, provided more sophisticated models are available. Certainly, with the time needed for validation programmes, the development of modelling sophistication will not keep pace with the upgrades in computer performance. It is unlikely then, that system-code NRS analyses will ever again require super-computing power.

However, even with the advances in computer technology, it is difficult to see CFD codes being capable of simulating the whole primary or secondary loop of a nuclear plant: system and component codes will still remain the main tools for this. However, for those occasions when CFD is needed – and many examples of this have been given in this document – the computations will stretch computing resources to the limit, just as predicted by Parkinson’s Law.

The CFD codes will allow the zooming in on specific zones of a circuit, or may be used as a tool to derive new closure relations for more macroscopic approaches, reducing the necessity of expensive experimental programmes. Coupling between CFD and system codes may also be an efficient way to improve the description of small-scale phenomena, while living within current computer limitations. As soon as in-progress developments are available, Direct Numerical Simulation (DNS) codes will be used for a better understanding of small-scale physical processes, and for the derivation of new models for averaged approaches.

These days, CFD simulations using 10 million nodes are common in many industrial applications. Such computations are possible because invariably the calculations are steady-state, single-phase, and carried out using parallel-architecture machines. In NRS applications, many of the situations requiring analysis are of a transient nature. CFD codes are computationally demanding, both in terms of memory usage and in the number of operations. Since the accuracy of a solution can be improved by refining the mesh, and by shortening the time step, there is a tendency to use whatever computational resources are available, and there is a never-ending and never-compromising demand for faster machines and more memory – Parkinson’s Law again!

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 limited, remains, for purely practical reasons, roughly proportional to 1/N, so the number of time steps is also proportional to N. Present-day commercial CFD codes are still based on a pressure-velocity coupling algorithm, which entails the iterative solution of a large linear system of equations. Much of the CPU overhead (sometimes up to 90%) derives from this procedure. Typically, the number of iterations M to convergence within a time step is also proportional to N. Thus finally, the run-time for the CFD code should scale according to

$$t \propto N^5$$

where the constant of proportionality, among other things, depends linearly on the total simulation time – and simulation times in NRS applications can be very long.

Despite the continual improvement in processor power, the commodity computer market has still not overtaken the demands of CFD. Traditionally, programs were written to run on a single processor in a serial manner, with one operation occurring after the next. One way to achieve a speed-up is to divide up the program to run on a number of processors in parallel, either on a multiprocessor machine (a single computer with multiple CPUs), or on a cluster of machines accessed 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 (Ref. 3) has shown that the scaling up of performance with number of processors is strongly dependent on the size of the system arrays (i.e. number of meshes), as well as on the details of the computer architecture and memory hierarchy. The speed of a program also depends on the language (generally, Fortran is faster than C), the compiler (levels of optimisation), and the syntax used to express basic operations (machine-dependent). With regards to the syntax of operations, forms that are fast on one platform might be slow on another. Modern workstations have proved to give good performance for small array sizes that fit into the processor's cache. However, when the array is too large to fit into the cache, the speed of the computers can drop to half their peak performance. These machines commonly bank their memory, and array sizes, which results in the same memory bank being accessed multiple times for the same operation, and will incur a performance penalty as a result. This problem can commonly be solved by increasing the leading dimension of an array.

Vector computers have an optimum speed when the array dimensions are a multiple of the size of the vector registers, typically a multiple of 8. Thus, when comparing a vector computer to a workstation, the optimum array size for the vector platform is the slowest (due to memory banking) on the workstation. Shared memory parallel computers typically give good performance for small to moderate problem sizes, for which the data fits within the cache of the computer's processors, but if array sizes are too large for the data to fit into the cache, there is a severe drop in speed, as all processors attempt to access the shared memory. In comparison, it was found (Ref. 3) that distributed memory machines achieved poor speeds for small to moderate array sizes, whereas for large problems, for which the memory access speed rather than inter-processor communication speed dominated, the parallel paths to memory ensured a near linear speedup with number of processors.

Given this linear speed-up, and the N^5 dependence of runtime on number of meshes in one coordinate direction, doubling the number of processors, and keeping total runtime the same, the number of meshes in each direction can be increased by about 15%, say from 100 to 115. Conversely, doubling the mesh density, say from 100 to 200 in each coordinate direction, again keeping total runtime constant, means that the number of processors has to be increased by a factor 32.

Given the above 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, 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 entire transient. Certainly, expanding efforts in NRS will ensure that Parkinson's Law will prevail for CFD.

Ref. 1: C. Northcote Parkinson, *Parkinson's Law: The Pursuit of Progress*, London, John Murray (1958).

Ref. 2: M. Livolant, M. Durin, J.-C. Micaeli, "Supercomputing and Nuclear Safety", Int. Conf. on Supercomputing in Nuclear Applications, SNA'2003, Paris, Sept. 22-24, 2003.

Ref. 3: S. E. Norris, "A Parallel Navier-Stokes Solver for Natural Convection and Free-Surface Flow", Ch. 6, PhD Thesis, Dept. Mech. Eng., University of Sydney, Sept. 2000.

6.11 Special Considerations for Liquid Metals

Relevance of the phenomena as far as NRS is concerned

The conventional fast breeder reactor uses liquid metal, such as Na, NaK or Pb etc., as coolant. The following liquid-metal hydraulics phenomena are relevant as far as NRS is concerned: (i) natural convection, (ii) thermal striping, (iii) sloshing of free surface, (iv) sodium fires, and (v) sodium boiling. It seems that some established CFD studies have been carried out concerning natural convection and sodium fires; these are described in Section 3.22 of this report. Identification of gaps in the technology base for special considerations for liquid metals, therefore, is restricted to thermal striping, sloshing of the free surface and sodium boiling.

What the issue is

Thermal striping phenomena in LMFBRs, characterised by stationary, random temperature fluctuations, are typically observed in the region immediately above the core exit, and are due to the interaction of cold sodium flowing out of a control rod assembly and hot sodium flowing out of adjacent fuel assemblies. The same phenomenon occurs at a mixing tee, a combining junction pipe, etc. The temperature fluctuations induce high-cycle fatigue in the structures.

The sodium in the reactor vessel has a free surface, and is covered by an inert gas. When the reactor vessel is shaken by seismic forces, waves will form on the free surface: the so-called "sloshing behaviour". If the amplitude of the wave increases, the inert gas may enter an inlet nozzle and be carried around the primary circuit, resulting in the formation of gas bubbles in the core region, causing a positive reactivity insertion. Another issue is the fluid force associated with slug movement caused by violent sloshing. The vessel wall and internal structures of LMFBRs are relatively thin, and mitigate thermal stress attributed to temperature variations during operation, which is characteristic of the high conductivity of liquid sodium. The fluid force of a moving liquid slug, therefore, could threaten the integrity of the reactor vessel.

Sodium boiling in the core region of LMFBRs would cause a power excursion, through feedback of positive reactivity coefficient of sodium void.

What the difficulty is and why CFD is needed to solve it

The design study associated with the protection of the Japanese LMFBR MONJU from thermal striping was performed using experimental data from a 1/1 scale model with sodium. In such a conventional approach, an increase in costs, as well as the time to perform the experiments, is inevitable, because it is technically difficult to obtain adequate amounts of quality of data from sodium experiments. CFD is needed to overcome this difficulty.

Linear-wave theory is applicable only to small-amplitude waves at the free surface. CFD is needed to solve the (non-linear) violent sloshing phenomenon important for NRS.

High accuracy is required from the sodium-boiling model, whose function is first to predict the exact time and location of the onset of boiling, and then to describe the possible progression to dryout. CFD has the potential to improve the accuracy in prediction of these phenomena.

What has been attempted and achieved / what needs to be done (recommendations)

The IAEA coordinated a benchmark exercise with the goal of simulating an accident in which thermal striping had caused a crack in a secondary pipe of the French LMFBR Phenix. JNC has been developing a simulation system for the thermal striping phenomena consisting of two CFD codes: AQUA and DINUS-3. AQUA is a 3D model for porous media with a RANS turbulent model, and DINUS-3 is a 3D model for open medium, with a DNS turbulent model (see Ref. 1).

There are two approaches being used to simulate free surface flows numerically. One assumes potential flow conditions, in which the basic equations to be solved are the Bernoulli equation with a velocity potential, the kinematical equation of the liquid surface, and the mass conservation equation of the liquid (see Ref. 2). The other uses a commercial CFD code that incorporates the VOF interface-tracking technique (see Ref. 3).

Numerous out-of-pile and in-pile experiments have been conducted to obtain information on sodium boiling, because in the past the power excursion scenario due to positive feedback of sodium void received the most attention by the LMFBR safety community. Whole-core accident analysis codes, such as SAS4A (see Ref. 4), have been developed for this purpose: they use a one-dimensional approach for the sodium-boiling module.

Ref. 1: T. Muramatsu et al., “Validation of Fast Reactor Thermomechanical and Thermohydraulic Codes”, Final report of a coordinated research project 1996-1999, IAEA-TECDOC-1318, 2002.

Ref. 2: M. Takakuwa et al., “Three-Dimensional Analysis Method for Sloshing Behavior of Fast Breeder Reactor and its Application to Uni-vessel Type and Multi-vessel Type FBR”, Proc. Int. Conf. on Fast Reactors and Related Fuel Cycles, Vol. I, Oct. 28-Nov. 1, 1991, Kyoto, Japan.

Ref. 3: Seong-O. Kim et al., “An Analysis Methodology of Free Surface Behavior in the KALIMER Hot Pool”, Proc. Third Korea-Japan Symposium on Nuclear Thermal Hydraulics and Safety, Oct. 13-16, 2002, Kyeongju, Korea.

Ref. 4: H.U. Wider et al., “Status and validation of the SAS4A accident code system”, Proc. Int. Topical Meeting on LMFBR Safety and Related Design and Operational Aspects, Vol. II, p.2-13, Lyon, 1982.

6.12 Scaling and Uncertainty

6.12.1 The scaling issue

The word scaling can be used in a number of contexts: two of these may be listed here.

1. Scaling of an experiment is the process of demonstrating how and to what extent the simulation of a physical process (e.g., a reactor transient) by an experiment at a reduced scale (or at different values of some flow parameters, such as pressure and fluid properties) can be sufficiently representative of the real process in the reactor.
2. Scaling applied to a numerical simulation tool is the process of demonstrating how and to what extent the numerical simulation tool validated on one or several reduced scale experiments (or at different values of some flow parameters, such as pressure and fluid properties) can be applied with sufficient confidence to the real process.

One should emphasise that scaling is meant here in terms of the prediction of a result for the reactor from a scaled experiment, as defined in Oberkampf & Roy in their book on V&V (2010).

When solving a reactor thermal-hydraulic problem, the answer to the issue may be:

1. Purely experimental: the experiments can tell what would occur in the reactor with sufficient accuracy and reliability
2. Purely numerical: only numerical simulations are used to solve the problem
3. Both experiments and simulation tools are used to solve the issue.

The first case is not common, and is not considered here since CFD simulation tools are not involved. The second case is also not common, due to the limited reliability and accuracy of thermal-hydraulic simulation tools. So we will focus here on the third case, in which both experiments and simulation tools are used to try to resolve the issue. This means that the simulation tool is used to extrapolate from experiments to the reactor situation, and that the degree of confidence in this extrapolation is itself part of the scaling issue.

The extrapolation to a reactor situation made by a single-phase CFD tool introduces several new aspects, and raises several questions:

- How to guarantee that a CFD code can extrapolate from a reduced-scale validation experiment to the full-scale application?
- How to extrapolate nodalisation from a reduced-scale validation experiment to the full-scale application?
- How to extrapolate:
  - from one fluid to another?
  - to a different value of the Re number and/or to a different value of any other non-dimensional number important in the physical processes taking place?

In any case, numerical simulation of scaled experiments has a given accuracy defined by the error on some target parameters, and one should determine how the code error changes when extrapolating to the reactor situation.

Therefore, scaling associated with a CFD application is part of the CFD code uncertainty evaluation, and is a necessary preliminary step in this uncertainty evaluation.

Both scaling and uncertainty are closely related to the process of Validation and Verification. The definition of a metric for the validation is also part of the issue.

6.12.2 The scaling methodologies

6.12.2.1 General problems of scaling

Scaling analyses address the following question: how experimental results can be transferred from experimental conditions to prototype conditions if differences exist with respect to the following parameters:

i. Geometrical dimensions, power and shapes (e.g., small-scale experiments)

ii. The choice of materials (e.g., helium instead of hydrogen in the atmosphere, artificial aerosols BiO2 instead of Sr or Cs)
iii. Time scales (e.g., accelerated thermal ageing), or material loads (e.g., artificial irradiation sources).

In order to transfer experimental results to prototype conditions, the experimental data are often condensed in the form of correlations for use in a numerical code. These correlations are expressed as relations among non-dimensional pi-monomials, but what pi-monomial should be selected in order to scale a given magnitude correctly to prototype conditions?

In this case, the structure of the code variables must be taken into account. Generally, codes are formulated in terms of local coordinates; this means that introduction of non-local interaction terms (e.g. heat transfer correlations with local coordinate dependency, such as the distance from the entrance of the pipe) are difficult to implement.

Also, the correlations for lumped-parameter codes may be quite different from the corresponding correlations for CFD codes. For instance, two-phase heat transfer correlations for a 1D channel in TH-codes depends in general on average channel magnitudes, and are not applicable to CFD codes. Frequently, a result of a scaling analysis is a scale-independent correlation that is derived from experiments and is often implemented in a computer code for simulating some phenomena, like heat transfer, or condensation rate.

6.12.2.2 General methodology on scaling H2TS

For application in nuclear reactor safety, a comprehensive methodology named H2TS (“Hierarchical Two Tiered Scaling”) was developed by a Technical Program Group of the U.S. NRC under the chairmanship N. Zuber. This work provided a theoretical framework and systematic procedures for carrying out scaling analyses. The name is based on using a progressive and hierarchised scaling methodology, organised in two basic steps. The first one is a top-down (T-D) approach and the second a bottom-up (B-U) approach.

The first step (T-D) is organised at the system or plant level, and is used to deduce non-dimensional groups obtained from the mass (M), energy (E) and momentum (MM) conservation equations, derived from the systems that have been considered as important according to a Phenomena Identification and Ranking Table (PIRT) exercise. These non-dimensional groups are used to establish the scaling hierarchy; i.e., what phenomena have priority in order to be scaled, and to identify what phenomena must be included in the bottom-up analysis.

The second part of the H2TS methodology is the B-U analysis itself. This is a detailed analysis at the component level, performed in order to assure that all relevant phenomena are properly represented in the balance equations that govern the evolution of the main variables in the different control volumes.

Most important steps to perform in the scaling analysis

This step consists of decomposition of the plant or system using the following hierarchy:

1. Systems (S): i.e., coolant system of a PWR.
2. Sub-systems (SS): RPV, accumulators, PRZ, RCP, SG.

3. Modules or Components (M): e.g., for the RPV, the main components are the downcomer, the reactor core, the lower plenum and upper plenum.
4. The components are divided in its constituents (C): e.g., the SG is divided into the 1-Φ tube side, the internals and the 2-Φ side.
5. The constituents are divided into phases (P): gas, liquid or solid.
6. Each phase can adopt different geometrical configurations (G): e.g., the liquid phase can be in the form of drops, liquid in the bulk, or liquid on the walls (condensate).
7. Each geometrical configuration is described by three conservation equations: Mass (M), Energy (E) and momentum (MM).
8. Finally, each conservation equation can be attributed different transfer processes.

II. The second step of the scaling analysis is to identify the scale level at which we must develop the similarity criteria. This is determined by the phenomena to be considered.

III. Once we have identified the scaling level, we must define all the control volumes and flow paths (convective and diffusive) connecting the identified control volumes (CV) of the system. Then we set the conservation equations in each CV previously identified (M, E, MM) and non-dimensionalise these conservation equations. After non-dimensionalisation, of the terms of the conservation equations, we will notice that they appear multiplied by groups of pi-monomials, known as Ξ groups. These groups can be expressed in terms of a minimum set of pi-monomials for each specific problem. Comparing the values of the different Ξ groups that appear in a given equation, we can assess the relative importance of each individual transfer process that contributes to a given conservation equation in a given CV.

IV. It is from these groups of pi-monomials that we deduce the scaling relations between the model and the prototype, and the distortions.

From simple to complex cases of scaling

The classical methods of dimensional analysis normally valid for simple non-interacting systems aim to produce the non-dimensional numbers that control a given phenomenon. These methods are usually applicable to relatively simple situations or single phenomena (such as heat transfer or frictional pressure loss), where the length and time scales of the problem are rather unique, and well-defined.

The classical, well-established methods are:

i) Use of the Buckingham Pi theorem: i.e., combination of all relevant variables to form dimensionless groups.
ii) Dimensionless numbers from known governing equations.
iii) To form dimensionless numbers as ratios of “competing quantities”, like force balances (for instance the Reynolds number formed as the ratio of the inertial to viscous forces).
iv) Dimensionless numbers as ratios of characteristic times for exchange of mass, energy and momentum over specified areas and volumes.

In analysing complex systems, where several phenomena interact at different spatial and time scales, one faces difficulties in applying the classical methods, since the multiplicity of scales results in too many non-dimensional numbers that cannot be assigned identical values, and therefore all the similarity conditions cannot be satisfied simultaneously.

In a certain number of scaling analyses, computer codes have been used. This may yield useful results in some cases, but codes rely on certain closure relations, and the scaling of these correlations must be assured. For example, a well validated code capable of spanning a range of scales could conceivably be used to simulate the behaviour of scaled facilities, verify the adequacy of the scaling and quantify the distortions. However, if one had sufficient faith in the predictions of a code at different scales, then tests at reduced scales and scaling analyses would not be needed. But this does not seem to be the case.

6.12.2.3 Fractional Scaling Analysis

The fractional scaling analysis method originated during the course of a program designed to scale severe accidents (Zuber 1991). For the purposes of thermal hydraulics, the information entities of interest in Zuber terminology are mass, momentum and energy, and the agents of change are fluxes of mass, momentum and energy across the system boundaries.

Fractional scaling is based on the integral approach, given that the interest is in spatial–temporal scaling of a system; that is, an aggregate of interacting components. Furthermore, the integral formulation has the following additional attributes:

1. it addresses and quantifies changes of a state variable within and around a finite region of space;
2. it is applicable to an aggregate of interacting components;
3. it introduces in the scaling analysis the initial and/or boundary conditions of interest to a specific problem;
4. it allows the inclusion of two important concepts ─ turnover time and turnover length; as an example, the first for a given volume V is defined as the inverse of the replacement frequency ω_τ ;
5. the path integrals introduce in the scaling analysis the concept of action, which relates the initial energy and the turnover time.

Fractional scaling is used to provide a synthesis of experimental data to generate quantitative criteria for assessing the effects of various designs and operating parameters on thermal-hydraulic processes in a nuclear power plant (NPP). The synthesis via fractional scaling is carried out at three hierarchical levels: process, component and system. The fractional scaling analysis (FSA) identifies dominant processes, ranks them quantitatively according to their importance, and provides thereby an objective basis for establishing phenomena identification and ranking tables (PIRTs) as well as a basis for conducting uncertainty analyses.

Consider a region of space referred to as the module M, characterised by a state variable SV, undergoing a change caused by an agent denoted by Φ, then one writes:

$$\frac{dSV}{dt} = \Phi .$$

Zuber defines the fractional rate of change (FRC) of this state variable SV as:

$$\omega = \frac{1}{SV}\frac{dSV}{dt} = \frac{\Phi}{SV} = \frac{\text{Cause}}{\text{Effect}} = \frac{1}{\tau} .$$

The FRC is the inverse of the characteristic time for the process causing the change.

If we have several agents Φ1,Φ2… causing the change, then the fractional rates of changes FRCs quantify the intensity of each process (agent of change) affecting the state variables in terms of what fraction of the variable total change the agent was responsible for. The spatial scale (characteristic length) for a magnitude being transferred across an area A and integrated (felt) within a volume V, is given by the inverse of the transfer area concentration A/V.

Two time scales are assigned to each module M, the first is the “clock” time; the time during which the change is being observed. The second is the “process” time τ that characterises the change of a state variable SV caused by a particular agent of change. Two or more interacting modules, each having its own state variable SV, form an aggregate, and can be modelled as an aggregate-module characterised by an effective state variable. Also, it is possible to have a module M with a state variable acted upon by two agents $\Phi 1$ and $\Phi 2$.

Another important element of fractional scaling analysis is the effect metric Ω, which quantifies the effect that the agent of change Φ has on one state variable during a period of time δt, and is given by $\Omega = \omega \delta t$. Consequently, processes having the same effect metric will be similar because their state variables have been changed by the same fractional amount.

The application of FSA to NPPs can be structured by addressing the problem at three hierarchical levels, process, component and system. According to Zuber, at each hierarchical level one considers questions of increasing complexity:

- At the process level, the question is what is the effect on the change of the corresponding state variable?
- At the component level the questions are: given a component, what are the effects of various processes on the change of a state variable? What is the ranking of their importance in that change? What are the effects of scale distortions in geometry and/or time on the change of a state variable?
- At the system level, the questions are: given a system and a postulated TH scenario, what are the governing processes and the corresponding components? What is the ranking of their importance on the postulated TH process? What are the effects of the component distortions, if present? What are the component interactions?

The purpose of applying FSA to a NPP is to develop a method that can address all these questions at all levels of interest. The application of FSA is structured at the three levels mentioned earlier: process, component and system.

- At the process level, a synthesis of the parameters governing a particular process is achieved through the effect metrics Ω.
- At the component level the synthesis is performed on process via the effect metrics Ω. When several processes act together to change a state variable in a given component, the effect of each one is quantified by the corresponding effect metric Ω. In this way ordering the effect metrics by their magnitudes generates the hierarchy of processes, i.e. it ranks the importance of the processes by their change in a given component. Therefore, this level produces quantitative criteria for identifying governing processes that must be addressed in computations and experiments. For code developers, this process hierarchy provides rational guidance and justification for simplifying computer models and for concentrating on the important processes. For experiments, it establishes scaling priorities.

At the system level, the synthesis is performed via the system matrix, which combines the components as rows with their processes as columns. For a given component, the associated row ranks the effect of each process by the Ω as a percentage change of a given quantity. For a given process, the column ranks the effect of that process on each component according to its Ω_j.

6.12.2.4 Examples of scaling analyses for an experiment

According to Wulff (1996), the purpose of scaling analyses is to provide:

1. the design parameters for reduced-size test facilities;
2. the conditions for operating experiments, such that at least the dominant phenomena taking place in the full-size plant are reproduced in the experimental facility over the range of plant conditions;
3. the non-dimensional parameters that facilitate the efficient and compact presentation and correlation of experimental results, which, by virtue of similarity and the parameter selection, apply to many systems, including both the test facility and the full size plant;
4. to identify the dominant processes, events, and characteristics (properties), all called here collectively "phenomena", to substantiate quantitatively, or revise, the expert-opinion-based, but still subjective, ranking of phenomena in the order of their importance, i.e. the ranking which is normally arranged in the Phenomena Identification and Ranking Table (PIRT);
5. to select among all the available test facilities the one that produces optimal similarity and the smallest scale distortion, and to establish thereby the test matrix;
6. to provide the basis for quantifying scale distortions; and
7. to derive the scaling criteria, or simulating component interactions, within a system from the global component and system models, with the focus on systems, rather than component scaling.

Traditional scaling analyses embody first normalizing the conservation equations on the subsystem or component level for the test section, then repeating this subsystem level scaling for all the components in the system, and collecting all the local scaling criteria into a set of system scaling criteria. The claim is then made that the dynamic component interaction and the global system response should be scaled successfully with the set of criteria for local component scaling, because the system is the sum of its components. This principle applies only if all the local criteria are met, and complete similitude exists. Complete similitude, however, is physically impossible, because all scaling requirements cannot be met simultaneously for a system in which areas and volumes, and, therefore area-dependent transfer rates and volume-dependent capacities, scale with different powers of the length parameter, and thereby produce conflicting scaling requirements.

Scaling groups can be derived using several methods, but two fundamental principles of scaling must be met (Wulff, 1996):

- the governing equations are *normalised* such that the normalised variables and their derivatives with respect to normalised time and space coordinates are of order unity, and the magnitude of the normalised conservation equation is measured by its normalising (constant) coefficient;
- the governing equations are then *scaled* by division through by the coefficient of the driving term; this renders the driving term of order unity, and yields fewer non-dimensional scaling groups, which measure the magnitudes of their respective terms, and therewith the importance of the associated transfer processes, relative to the driving term.

A categorisation of scaling approaches can be found, e.g., in Yadigaroglu & Zeller (1994).

- The simplest scaling technique is linear scaling, in which all length ratios are preserved: the mass, momentum and energy equations of a system, along which the equation of state, are non-dimensionalised, and scaling criteria are then derived from the resulting parameters; linear scaling leads to time distortion.

- Volumetric or time-preserving scaling is another frequently used technique, also based on scaling parameters coming from the non-dimensionalised conservation equations; models scaled by this technique preserve the flow lengths, while areas, volumes, flow rates and power are reduced proportionally.
- Time-distorted scaling criteria, described e.g. in Ishii & Kataoka (1984), include both linear and volumetric scaling as special cases, see Kiang (1985).
- A "structured" scaling methodology, referred to as hierarchical two-tiered scaling (H2TS), and proposed by Zuber (see e.g. Zuber, 1999), addresses the scaling issues in two tiers: a top-down (inductive) system approach, followed by a bottom-up, process-and-phenomena approach, since traditional local and component-level scaling cannot produce the scaling criteria for component interaction.

The last approach is described, e.g., also in Zuber et al. (1998) and Wulff (1996), but its principles and procedures can be best made clear by its application to design of the APEX test facility (Advanced Plant Experiment, Oregon State University), see Reyes & Hochreiter (1998). A short summary of their analysis follows. The objective of this scaling study was to obtain the physical dimensions of a test facility that would simulate the flow and heat transfer during an AP600 Small Break LOCA. The APEX scaling analysis was divided into four modes of operation, each corresponding to a different phase of the SBLOCA:

- closed loop natural circulation;
- open system depressurization;
- venting, draining and injection;
- long-term recirculation.

For each mode of AP600 safety system operation, the following specific scaling objectives were met:

- the similarity groups, which should be preserved between the test facility and the full-scale prototype, were obtained;
- the priorities for preserving the similarity groups were established;
- the important processes were identified and addressed;
- the dimensions for the test facility design, including the critical attributes, were specified; and
- the facility biases due to scaling distortions were quantified.

To achieve this, eight tasks had to be performed during the scaling analyses.

- To specify experimental objectives.
- To prepare the SBLOCA Plausible Phenomena Identification and Ranking Tables (PPIRTs) for each of the phases of a typical SBLOCA transient. Existing data on standard PWRs, coupled with engineering judgment and calculations for the AP600, were used to determine which SBLOCA thermal-hydraulic phenomena might impact core liquid inventory or fuel peak clad temperature.
- H2TS analysis for each phase of the SBLOCA was performed. The four basic elements of the H2TS method are:
  - *System subdivision*. The AP600 was subdivided into two major systems: a reactor coolant system and a passive safety system. These systems were further subdivided into interacting subsystems

(or modules), which were further subdivided into interacting phases (liquid, vapour or solid). Each phase was characterised by one or more geometrical configurations, and each geometrical configuration was described by one or more field equations (mass, energy and momentum conservation equations).

- *Scale identification*. The scaling level (system level, subsystem level, component level, constituent level) depending on the type of phenomena being considered was identified. A set of control volume balance equations was written for each hierarchical level.
- *Top-down scaling analysis*. For each hierarchical level, the governing control volume balance equations were written and expressed in dimensionless form by specifying dimensionless groups in terms of the constant initial and boundary conditions. Numerical estimates of the characteristic time ratios, Π_k, were obtained for the prototype and the model for each phase of the transient at each hierarchical level of interest. Physically, each characteristic time ratio is composed of a specific frequency, ω_k, which is an attribute of the specific process, and the residence time constant, τ_k, for the control volume. The specific frequency defines the mass, momentum or energy transfer rate for a particular process. The residence time defines the total time available for the transfer process to occur within the control volume. If $\Pi_k<<1$, only a small amount of the conserved property would be transferred in the limited time available for the specific process to evolve, and the specific process would not be important to the phase of the transient being considered. On the other hand, if $\Pi_k \geq 1$, the specific process evolves at a high enough rate to permit significant amounts of the conserved property to be transferred during the time period τ_k.
- *Bottom-up scaling analysis*. This analysis provided closure relations for the characteristic time ratios. The closure relations consisted of models or correlations for specific processes. These closure relations were used to develop the final form of the scaling criteria for purposes of scaling the individual processes of importance to system behaviour.

- The scaling criteria were developed by setting the characteristic time ratios for the dominant processes in the AP600 to those for APEX at each hierarchical level.
- The effect of a distortion in APEX for a specific process was quantified by means of a distortion factor DF, which physically represents the fractional difference in the amount of conserved property transferred through the evolution of a specific process in the prototype to the amount of conserved property transferred through the same process in the model during their respective residence times. A distortion factor of zero means that the model ideally simulates the specific process.
- System design specification. The outcome of the scaling analysis was therefore a set of characteristic time ratios (dimensionless Π groups) and similarity criteria for each mode of operation. These scaling criteria were expressed in terms of ratios of model to prototype fluid properties, material properties, and geometrical properties. Now, working fluid, component materials, operating pressure, and the length, diameter and time scales can be selected.
- Evaluation of key T/H PPIRT processes to prioritise system design specification.
- APEX test facility design specifications and Q/A critical attributes.

Recently, Yun et al. (2004) developed a new approach, called the modified linear scaling method, from the incompressible, two-dimensional, two-fluid model for an annular and annular-mist flow patterns without a priori considering the interfacial heat transfer. In the dimensionless governing equations, the aspect ratio of the downcomer (the ratio between a height and a lateral length of downcomer) was preserved as in a prototype, and the velocity of each phase was normalised by introducing the Wallis parameter, which means the ratio between the inertia force and the gravitational force. The dimensionless parameter was also used for the analysis of the UPTF Test 21D (MPR-1329, 1992) and it is defined as follows;

$$j_k^* = \alpha_k u_k \left[\frac{\rho_k}{(\rho_f - \rho_g) \cdot g \cdot d} \right]^{1/2}$$

The scaling criteria required for the modified linear scaling method are listed in the Table below, where they are also compared with those for the standard linear scaling method.

Table: Comparison of the scaling methodologies

Parameter	Scaling Ratio	
	Linear Scaling	Modified Linear Scaling
Length Ratio, l_R	l_R	l_R
Area Ratio, a_R	l_R^2	l_R^2
Volume Ratio, V_R	l_R^3	l_R^3
Time Ratio, t_R	l_R	$l_R^{1/2}$
Velocity Ratio, υ_R	1	$l_R^{1/2}$
Flow Rate Ratio, $\dot{m}_R$	l_R^2	$l_R^{5/2}$
Pressure Drop Ratio, Δp_R	1	l_R
Gravity Ratio, g_R	l_R^{-1}	1
Pressure Ratio, p_R	1	1
Temperature Ratio, T_R	1	1
Void Ratio, α_R	1	1
Slip Ratio, S_R	1	1
Aspect Ratio, l_R / D_R	1	1

The present scaling method requires the same geometrical similarity as in the case of the standard linear scaling method, whereas the flow velocity for steam and ECC water should be scaled in the form of the Wallis type of a dimensionless velocity. In this scaling method, the velocity and time scales are reduced according to the square root of the length scale. This naturally leads to preserving the gravity effect on the flow phenomena even in the scaled tests.

The subject of scaling is very broad and cannot be dealt with in depth in this document. For CFD applications to NRS, it is comforting that, in principle, the computational model can be at 1-1 scale, but it remains important to ensure that the fluid-dynamic phenomena of relevance, validated against scaled experiments, have been preserved. This may be difficult if the fluid behaviour is categorised by flow-regime maps.

6.12.2.5 Example of non-dimensional analysis applied to CFD Codes

Each term in the conservation equations is associated to a physical process, and each one of these processes has inherent length and time scales. One of the most important tools for determining the relative magnitude of the various terms, and in this way to reduce the number of true parameter in the equations, is through non-dimensional scaling analysis.

There are three are main objectives of the non-dimensional analysis applied to CFD codes. The first is to know the non-dimensional numbers, such as Reynolds number Re, Prandtl number, Schmidt number, and so on, that govern the solution of the given problem. The second one is to understand the relative magnitudes of the various processes that contribute to a given conservation equation, and to reduce the number of true parameters in the equations. The third is to make the equations more tractable for numerical solution once all the non-dimensional variables have the same order.

The first step is therefore the conversion of the instantaneous conservation equations to their non-dimensional forms in order to see the dependence of these equations with the classic non-dimensional numbers for different situations. The non-dimensionalisation of the conservation equations in Cartesian coordinates can be performed in different ways. To show the physical sense of all the terms is better to define the non-dimensional magnitudes as follows:

$$u_i^+ = \frac{u}{u_0},\ \ x_j = \frac{x_j}{L_0},\ t^+ = \frac{t}{t_0} = f_0\, t\ ,\ p^+ = \frac{p - p_\infty}{p_0 - p_\infty}\ ,\ \mu^+ = \frac{\mu}{\mu_0},\ \rho^+ = \frac{\rho}{\rho_0},\ g_j^+ = \frac{g_j}{g}, \tag{1}$$

where the sub-index 0 denotes the reference values for the problem, and p_0-p_∞ is the reference pressure difference.

Also we need to non-dimensionalise the boundary conditions: for instance, if we have an inlet boundary condition, we set:

$$\mathrm{u_{in,i}}^+ = \frac{\mathrm{u_{in,i}}}{\mathrm{u_0}} \tag{2}$$

The conservation equation for the i-th component of the momentum of an incompressible fluid is:

$$\frac{\partial}{\partial \mathrm{t}}\mathrm{u_i} + \mathrm{u_j}\frac{\partial \mathrm{u_i}}{\partial \mathrm{x_j}} = -\frac{1}{\rho}\frac{\partial \mathrm{p}}{\partial \mathrm{x_i}} + \nu\frac{\partial^2 \mathrm{u_i}}{\partial \mathrm{x_j^2}} + \mathrm{g_i} \tag{3}$$

That, after non-dimensionalisation yields:

$$\{\mathrm{St}\}\frac{\partial}{\partial \mathrm{t}^+}\mathrm{u_i}^+ + \mathrm{u_j}^+\frac{\partial \mathrm{u_i}^+}{\partial \mathrm{x_j}^+} = -\{\mathrm{Eu}\}\frac{\partial \mathrm{p}^+}{\partial \mathrm{x_i}^+} + \left\{\frac{1}{\mathrm{Re}}\right\}\nu^+\frac{\partial^2 \mathrm{u_i^+}}{\partial \mathrm{x_i^{+2}}} + \left\{\frac{1}{\mathrm{Fr}^2}\right\}\mathrm{g_i^+} \tag{4}$$

where we have used the standard definition of the Reynolds (Re), Strouhal (St), Froude (Fr) and Euler (Eu) numbers:

$$\mathrm{Re} = \frac{\rho_0 u_0 L_0}{\mu_0},\quad \mathrm{St} = \frac{f_o L_0}{u_0},\quad \mathrm{Fr} = \frac{u_0}{\sqrt{g L_0}},\quad \mathrm{Eu} = \frac{p_0 - p_\infty}{\rho_0 u_0^2} \tag{5}$$

In some flows, the boundary conditions define additional dimensionless numbers that do not appear explicitly in the conservation equations. Equation (4) is written in non-dimensional form, but is not necessary normalised. In order to normalise the equation properly, we need to choose the scaling parameters L0, u0, t0,…appropriately for the flow problem being analysed in such a way that all non-dimensional magnitudes, such as p+, t+, u+,… are of order of magnitude unity. Once we have normalised, the momentum conservation equations, we can compare the relative importance of the different terms in these equations by comparing the relative magnitudes of the coefficients of these terms, expressed in terms of well-known non-dimensional numbers.

We note in equation (4) that if all the physical magnitudes are properly normalised, then, if for instance for a given flow the Reynolds number is large, then advection dominates over the diffusion. If the Froude number is large, the gravity effects are negligible. In this way, we can know for a given problem what the most important terms are, and which terms can be neglected.

Let us turn our attention to the energy equation. In this case, for the sake of simplicity, we consider an incompressible flow with constant heat capacity, c_p, and we neglect the viscous heat generation and the compression work terms. With these simplifications the energy conservation equation can take the form:

$$\frac{\partial}{\partial \mathrm{t}}\rho \mathrm{c_p T} + \mathrm{c_p}\,\rho \mathrm{u_j}\,\frac{\partial \mathrm{T}}{\partial \mathrm{x_j}} = \left(\frac{\partial}{\partial \mathrm{x_j}}\mathrm{k}\frac{\partial}{\partial \mathrm{x_j}}\mathrm{T}\right) + \mathrm{S_h} \tag{6}$$

The non-dimensionalisation of this equation is performed using expressions (1) and (2) and the additional definitions:

$$k^{+}=\frac{k}{k_0};\qquad T^{+}=\frac{T-T_0}{\Delta T_0};\qquad S_h^0=\frac{S_h}{\rho_0 c_p \Delta T_0 u_0 / L_0};\tag{7}$$

where St is the Strouhal number, and $\Pr=\dfrac{c_{p0}\mu_0}{k_0}$ is the Prandtl number. This means that if the Peclet number (= Re×Pr) number is large, then advection dominates over diffusion in problems involving temperature change.

For the transport of a passive scalar a, with mass fraction Φ_a and concentration $C_a = \rho\varphi_a$ (kg/m^3), the conservation equation is given by:

$$\frac{\partial}{\partial t}\rho\Phi_a+\frac{\partial}{\partial x_j}\left(\rho\, u_j\, \Phi_a-\rho\,\Gamma\,\frac{\partial}{\partial x_j}\Phi_a\right)=S_a\tag{9}$$

where Γ is the diffusion coefficient. The non-dimensionalisation of this equation is performed defining the following non-dimensional variables:

$$\rho^{+}=\frac{\rho}{\rho_0},\ \Phi_a^{+}=\frac{\Phi_a}{\Phi_{a,0}},\ \Gamma_0^{+}=\frac{\Gamma}{\Gamma_0},\ S_a^{+}=\frac{S_a}{\rho_0\Phi_{a,0}(L_0/u_0)}\tag{10}$$

Then, on account of definitions (1) and (10), the conservation equation (9), for the concentration of a passive scalar a, can be recast in the form:

$$\{St\}\frac{\partial}{\partial t^{+}}\rho^{+}\Phi_a^{+}+\frac{\partial}{\partial x_j^{+}}\left(\rho^{+}\, u_j^{+}\, \Phi_a^{+}\right)-\left\{\frac{1}{\mathrm{Re\,Sc}}\right\}\frac{\partial}{\partial x_j^{+}}\rho^{+}\,\Gamma^{+}\,\frac{\partial}{\partial x_j^{+}}\Phi_a^{+}=S_a^{+}\tag{11}$$

where we have used the definition of the Schmidt number:

$$\frac{\Gamma_0}{L_0 u_0}=\frac{1}{\dfrac{\nu_0}{\Gamma_0}\,\dfrac{L_0 u_0}{\nu_0}}=\frac{1}{Sc_0\,Re_0}\tag{12}$$

In this case, the non-dimensional numbers that govern the importance of the diffusion process are the Reynolds and the Schmidt numbers.

We note that if all the physical magnitudes, geometric data, boundary conditions and source terms of a given problem are expressed in non-dimensional form, the solution of that given problem will be obtained by solving the non-dimensional conservations equations with the non-dimensional boundary conditions for that specific problem, and that two different problems with the same non-dimensionalised boundary conditions and geometric data in non-dimensional form will have the same non-dimensional solution, if it is verified that certain non-dimensional numbers are the same for the two problems. In this case, we can say that the two problems are similar. We note that this step is not required for the solution of a flow problem, because most of the CFD codes work with dimensional variables, but makes the problem set-up and subsequent analysis more convenient.

6.12.3 System code uncertainty methodologies

Code uncertainty methodologies for reactor thermal hydraulics were first developed for system codes, which simulate many kinds of transients in a very large range of single-phase and two-phase conditions. They were based on either propagation of the uncertainty of input parameters (so called uncertainty propagation methods) or accuracy extrapolation methods (see D'Auria & Galassi, 2010). But in other communities such as ASME, AIAA, marine hydrodynamics (F. Stern, et al,, 2001), other approaches adapted to CFD were more recently defined. These will be described in § 1.5. The system methods are described first.

The method using propagation of code input uncertainties follows the pioneering work of CSAU (NED Special Issue, 1990), later extended by GRS (Glaeser et al, 1994). It is the most often used method. Uncertain input parameters are first listed, including initial and boundary conditions, material properties and closure laws. Probability density functions (pdfs) are formulated for each input parameter. Then these uncertainties are propagated by running a reactor simulation using the system code. In the GRS method, a Monte Carlo approach is followed, with all input parameters being varied simultaneously according to their pdfs. The Wilks theorem is often used, which makes it possible to estimate the boundaries of the uncertainty range on any code response with a given degree of confidence. The number of code runs is around 100 for an acceptable degree of confidence, though a slightly higher number of code runs, typically 150 to 200, is advisable to have a better precision on the uncertainty ranges of the code response. The determination of the uncertainties of the closure laws can be made by simple engineering judgment, or better by some statistical approach, which use sensitivity methods and the results from many validation calculations (see de Crécy and Bazin, 2001-2004).

The method identified as propagation of code output errors is based upon the extrapolation of accuracy, i.e. UMAE (D'Auria & Debrecin, 1995) and CIAU (D'Auria & Giannotti).

Benchmarking of the two approaches was made within international projects launched by the OECD/CSNI. These are UMS (OECD/CSNI. 1998) and BEMUSE (de Crécy et al., 2007). These methods have now reached a reasonable degree of maturity, even if the quantification of the uncertainty of the closure laws remains a difficult issue.

The method using propagation of code input uncertainties require many calculations, which may be difficult in the context of CFD due to large required CPU times involved. Accuracy extrapolation methods require only one reactor simulation, but many preliminary validation calculations of Integral Test Facilities are required. The preliminary validation calculations are also required for propagation methods to determine the uncertainties of the closure laws if statistical methods are used. In this case, the calculated tests are Separate Effect Tests. In both propagation and extrapolation methods, the experimental uncertainties have to be taken into account.

6.12.4 Particularities of single-phase CFD applications

Many differences exist between the system codes, which solve mainly two-phase problems, and single-phase CFD tools:

- Single-phase CFD tools have very few physical models (turbulent viscosity, wall functions,…), whereas system codes include hundreds of closure laws for wall transfers and interfacial transfers for each flow regime, and for each flow geometry.
- Single-phase flow issues depend on a relatively small number of non-dimensional numbers compared to two-phase flow issues where many non-dimensional numbers may be involved. The scaling of a single-phase flow is more straightforward and more reliable than in two-phase situations for which many simplifying assumptions are often necessary.
- Single phase CFD tools propose many options for the physical models (k-ε, k-ω, RST, SST, RNG k-ε, LES, DES,…) whereas system codes generally propose one set of standard validated closure laws. No extended validation exists for each physical option.

- Single-phase CFD tools have many options available for the numerical scheme, whereas system codes generally propose just one (CATHARE, ATHLET, TRACE, SPACE) or two (RELAP-5, TRAC).
- Single-phase CFD tools do not propose a comprehensive validation matrix for each set of physical and numerical options, whereas system codes generally propose a very large validated matrix applied to a standard set of closure laws.
- Single-phase CFD tools may have CPU time difficulties to run simulations with a converged mesh and time step. Therefore, many applications may have significant numerical errors. This numerical error may be equal or larger than the error due to physical modelling. System codes may also use non-converged meshing, but generally the numerical error is much smaller than the error due to physical modelling, so that the latter may be ignored in the uncertainty analysis.
- Single-phase CFD tools are able to simulate the effects of small-scale geometrical details of the flow, whereas system codes are macroscopic tools which simplify the geometry of the flow and effects of small-scale geometrical details (e.g. geometry of spacer grids in a fuel assembly) are embedded in the closure laws which were fitted to data from prototypical experiments.

In summary, one can list the favourable and unfavourable aspects of scaling as an issue to be treated by single-phase CFD, compared to two-phase issues, as follows.

The favourable aspects are:

- Single-phase flow issues depend on a relatively small number of non-dimensional numbers. The scaling is straightforward and reliable, since it does not require many simplifying assumptions.
- Single-phase CFD tools have very few physical models the scalability of which has to be proven.
- The simplifications of the flow geometry for single-phase CFD tools are less frequent and less extreme than for system codes. Consequently, the portability of a physical model from a specific geometry to another one has not to be proven.

The unfavourable aspects are:

- When extrapolating from a scaled experiment simulation to a reactor simulation, the scalability of the numerical scheme and of the nodalization has to be investigated in addition to the scalability of the physical models.
- If CFD is used with some degree of simplification of the geometry, the impact of such simplifications should be taken into account in the scaling and uncertainty evaluation.
- Methodologies for scaling and uncertainty evaluation which would require many calculations would become very difficult in the context of CFD due to the high CPU cost of the calculations.
- Since several options for the physical models (turbulence, wall laws) and several numerical schemes are possible, if Best Practice Guidelines are not giving precise criteria to select the best choice of options, this represents an additional source of uncertainty which must be taken into account.
- Always for uncertainty evaluation, if a method of uncertainty propagation is chosen, quantifying the input uncertainties is a more complex issue than for a system code.

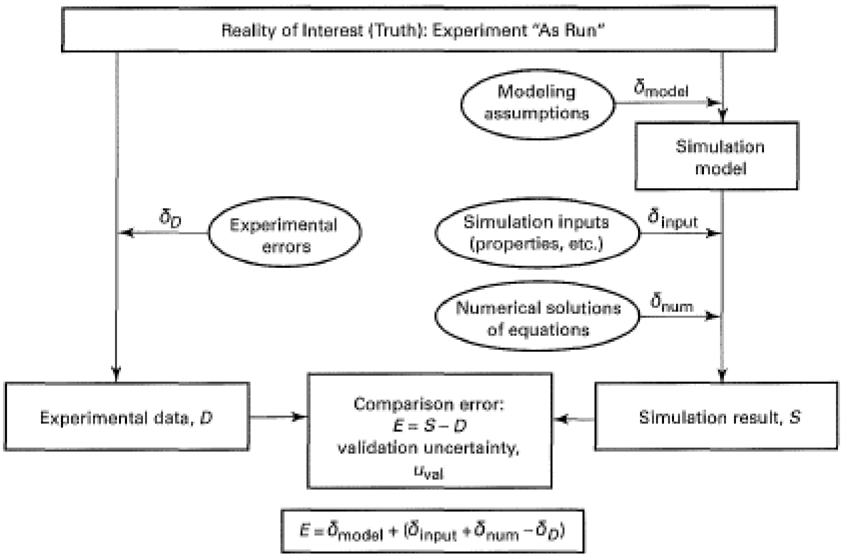
The absence of the results of a comprehensive validation matrix for single-phase CFD does not help in the scaling process.

6.12.5 *Existing CFD methods for uncertainty quantification*

ASME (American Society of Mechanical Engineers) has worked on a standard for verification and validation (V&V) and uncertainty qualification (UQ) for CFD and heat transfer applications (H.W. Coleman, 2009). The Standard conforms to US Nuclear Regulatory Commission (US NRC) and other regulatory practices, procedures and methods for licensing of NPPs, as embodied in the United States Code of Federal Regulations, and in other pertinent documents, such as Regulatory Guide 1.203: “Transient and Accident Analysis Methods”; and NUREG-0800: “NRC Standard Review Plan”.

This CFD standard is a part of V&V Standard Committee which includes three other standards on Integrated System Thermal Fluids behaviour (V&V 10), Solid Mechanics (V&V 30) and Medical Devices (V&V 40), as elaborated in E. A. Harvego, 2010). In practical terms, the standard V&V 20-2009 states that “The ultimate goal of V&V procedure is to determine the degree to which a model is an accurate representation of the real world”. This standard is strongly based on the use of experimental data for V&V and consequently for UQ. With this approach, ASME establish a strong link between V&V and UQ, in the same way as the methods described § 1.3, for which many preliminary validation calculations are required. Note that the V&V 20-2009 method is very much linked with the work of Oberkampf & Roy, 2010.

The global V&V-UQ process is outlines in the Table below. This Table only deals with uncertainties at the experimental scale. An additional term has to be evaluated for scaling from experimental to reactor scale.



A validation standard uncertainty, u_{val} can be defined as an estimate of the standard deviation of the parent population of the combination of errors ($\delta_{num} + \delta_{input} - \delta_D$):

$$\boxed{u_{val} = \sqrt{u_{num}^2 + u_{imput}^2 + u_D^2}}$$

The ASME standard gives solutions to evaluate every term of the validation error (E) and the validation uncertainty (u_{val}). Propagation methods are mainly used to evaluate uncertainties in input parameters. Uncertainties in the numerical solutions are given by the code verification step. This approach considers that experimental and numerical results of interest are scalars with uncertainties. Oberkampf & Roy (2010) describe a similar kind of methodology but for any kind of code results. Quantities of interest are considered as “p-box” entities, which are probability distributions considering epistemic uncertainties. Similar addition of terms is made to evaluate the code uncertainties, but specific mathematics for probability distributions are used. This approach can be more suitable for complex quantities of interest (for example CFD transient results).

Scaling uncertainty is not discussed in the ASME standard, but a chapter is dedicated to “prediction” in the work of Oberkampf & Roy. The main issue of error and uncertainties evaluation for scaling is that the "real" quantity of interest at reactor scale is generally unknown. One option is to use only code results to evaluate the scaling uncertainties. The main assumption then is to consider that the variation of code results between facility and reactor scale is equivalent to “real” variation between the scales. Another option is to use multiple experiments with a variation of scaling factors, like Reynolds or Froude Numbers. If available, a set of experiments can lead to the definition of a validation domain that contains the application domain, or gives some information for extrapolation outside of the validation domain.

The ASME standard methodology for uncertainty analysis underlines the role of V&V in the process of evaluating the confidence level of CFD code results. Uncertainties have to be evaluated step-by-step, using clearly defined numerical aspects of the code, such as time and space discretisation (i.e. time step and mesh convergence), or physical models (turbulence models, physical assumptions) with associated error evaluation.

The ASME committee intends to publish a supplement of this standard that will include an extension (as well as multivariate validation metrics). A presentation was made by P. Roache at the ASME 2012 V&V Symposium summarising the work in progress:

- The distinction between model quality vs. quality of the validation exercise.
- A brief review of interpolation vs. extrapolation curve-fitting, especially for high-dimensional parameter spaces.

This new version of the Standard V&V 20 is scheduled for release in 2012.

6.12.6 Some recommendations with regard to scaling associated to CFD applications

For solving a reactor safety issue by the application of single-phase CFD, several successive steps are necessary:

1. Scaling analysis is the first step. For this, the methods described above (H2TS and FSA) are recommended. These include:
    - Use of a PIRT to identify the dominant physical phenomena and their influence parameters to obtain a trustworthy analysis (see US-NRC Regulatory Guide 1.203)
    - The identification of the non-dimensional numbers that play an important role in the physical processes taking place; this is part of the PIRT analysis
    - The selection of relevant experimental data, or the definition of new experimental programs.
2. Selection of a CFD code and choice of the relevant physical and numerical options, and nodalisation; the choices should be made in accordance with Best Practice Guidelines, if they exist for this application. Otherwise, the choice among different numerical schemes and/or different physical models must be taken into account in terms of an uncertainty analysis.
3. Simulation of the relevant experimental data, as detailed elsewhere in this document.

4. Identification of the scale distortions of the available experimental data compared to the reactor application:
   - Are there different values of the non-dimensional numbers?
   - Are there significant differences in the geometry of the flow?
5. If an interpolation or an extrapolation from the values of the non-dimensional numbers of the experimental data to the values of the reactor application can give a sufficient confidence on the CFD simulation of the reactor case, the scalability of the CFD tool is good. This may be the case when the CFD simulations of several experiments having various values of the non-dimensional numbers are equally accurate. In case the scalability of the CFD is not evident, a quantitative evaluation of the CFD uncertainty has to be made either by accuracy extrapolation or by uncertainty propagation.
6. If there are significant differences in the geometry of the flow between the experiments and the reactor application, it should be demonstrated that this does not affect the accuracy of the simulation.
7. The numerical scalability of the CFD application should be considered for the whole process of experiments and reactor simulations. Several cases are possible:
   - if all experiments and reactor simulations were performed with converged time step and mesh size, and with full control of all numerical errors (see Best Practice Guidelines), there is no numerical scalability problem.

If some experiments or reactor simulations were performed with a non-fully-converged time step or mesh size, the numerical error should be estimated for each calculation to see if it is scale dependent. If necessary, such scale dependence should be taken into account in an uncertainty methodology. The Richardson method may be applied to estimate the numerical error.

Ref. 1: F. D'Auria, G.M. Galassi, "Scaling in nuclear reactor system thermal hydraulics", Nuclear Engineering & Design, doi:10.1016/j.nucengdes.2010.06.010, 2010.

Ref. 2: F. D'Auria, N. Debrecin, G.M. Galassi, "Outline of the Uncertainty Methodology based on Accuracy Extrapolation (UMAE)", Nuclear Technology, **109**(1), 21-38 (1995).

Ref. 3: F. D'Auria, W. Giannotti, "Development of Code with Capability of Internal Assessment of Uncertainty", Nuclear Technology, **131**(1), 159-196 (2000).

Ref. 4: S. Banerjee, M.G. Ortiz, T.K. Larson, D.L. Reeder, "Scaling in the safety of next generation reactors", Nuclear Engineering and Design, **186**, 111–133 (1998).

Ref. 5: H.K. Cho, et al., "Experimental Validation of the Modified Linear Scaling Methodology for Scaling ECC Bypass Phenomena in DVI Downcomer", Nuclear Engineering & Design, **235**, 2310-2322 (2005).

Ref. 6: H.K. Cho, et al., "Experimental Study for Multidimensional ECC Behaviors in Downcomer Annuli with a Direct Vessel Injection Mode during the LBLOCA Reflood Phase", J. of Nuclear Sci. & Technol., **42**(6), (2005).

Ref. 7: H.W. Coleman, et al., "Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer", ASME V&V 20-2009.

Ref. 8: A. de Crecy, P. Bazin, "Quantification of the uncertainties of the physical models of CATHARE 2", M&C 2001, Salt lake City, Utah, USA, September 2001, ANS Winter Meeting, Washington, DC, USA, Nov. 14-18, 2004.

Ref. 9: A. de Crécy, P. Bazin (Eds.) et al., Fujioka, Bemuse Phase III Report, Uncertainty and Sensitivity Analysis of the LOFT L2-5 Test, OECD/CSNI Report NEA/CSNI/R(2007)4, October 2007.

Ref. 10: E.A. Harvego, R.R. Shultz, R.L. Crane, "Development of a standard of verification and validation of software used to calculate nuclear system thermal fluids behaviour", ICONE 18-30243, May 17-21, 2010 Xi'an , China.

Ref. 11: M. Ishii, I. Kataoka, "Scaling laws for thermal-hydraulic system under single phase and two-phase natural circulation", Nuclear Engineering and Design, **81**, 411–425 (1984).

Ref. 12: M. Ishii, et al., "The three-level scaling approach with application to the Purdue University Multi-Dimensional Integral Test Assembly (PUMA)", Nuclear Engineering and Design, **186**, 177–211 (1998).

Ref. 13: K. Fischer, "Scaling of Containment Experiments (SCACEX)", FIR1-CT2001-20127, 2002.

Ref. 14: R.L. Kiang, "Scaling Criteria for Nuclear Reactor Thermal Hydraulics", Nuclear Science and Engineering, **89**, 207–216 (1985).

Ref. 15: R.F. Kunz, et al., "On the automated assessment of nuclear reactor systems code accuracy", Nuclear Engineering and Design, **211**(2-3), 245-272 (2002).

Ref. 16: S. Levy, *Two Phase Flow in Complex Systems*, Wiley and Sons.

Ref. 17: Nuclear Engineering and Design, Special Issue devoted to Scaling, 1998.

Ref. 18: Nuclear Engineering and Design, Special Issue devoted to CSAU, **119**, 1 (1990).

Ref. 19: W.L. Oberkampf, C.J. Roy, *Verification and Validation in Scientific Computing*, Cambridge University Press, 2010.

Ref. 20: P.F. Peterson, V.E. Schrock, R. Greif, "Scaling for integral simulation of mixing in large, stratified volumes", Nuclear Engineering and Design, **186**, 213–224 (1998).

Ref. 21: A. Petruzzi, F. D'Auria, F. "Approaches, Relevant Topics, and Internal Method for Uncertainty Evaluation in Predictions of Thermal-Hydraulic System Codes", J. Science and Technology of Nuclear Installations, Vol 2008, Art. ID 325071, 2008.

Ref. 22: A. Petruzzi, F. D'Auria, (Eds.), et al., "BEMUSE Programme. Phase 2 report: Re-Analysis of the ISP-13 Exercise, post test analysis of the LOFT L2-5 experiment", OECD/CSNI Report NEA/CSNI/R(2006)2, 1-625, 2006.

Ref. 23: V.H. Ransom, W. Wang, M. Ishii, "Use of an ideal scaled model for scaling evaluation", Nuclear Engineering and Design, **186**, 135–148, (1998).

Ref. 24: J.N. Reyes Jr., L. Hochreiter, "Scaling analysis for the OSU AP600 test facility (APEX)", Nuclear Engineering and Design, **186** 53–109 (1998).

Ref. 25: W. Schenk, "Scaling Analysis of Passive Containment Cooling Test", Inno Tepps (96) D004, January 1997.

Ref. 26: C.-H. Song et al., "Scaling of the Multi-Dimensional Thermal-Hydraulic Phenomena in Advanced Nuclear Reactors", Keynote Lecture, Proc. NTHAS5 (5th Korea-Japan Symp. on Nuclear Thermal Hydraulics and Safety), Jeju, Korea, Nov. 26- 29, (2006).

Ref. 27: F. Stern et al., "Comprehensive Approach to V&V of CFD. Part1: Methodology", ASME Journal of Fluids Engineering, **123**, 793-802 (2001).

Ref. 28: F. Stern et al., "Comprehensive Approach to V&V of CFD. Part 2: Application for RANS Simulation of a Cargo/Container Ship", ASME Journal of Fluids Engineering, **123**, 803-810 (2001).

Ref. 29: Scaling, Uncertainty and 3D Coupled Code Calculations in Nuclear Technology, J. STNI Special Issue, 2008.

Ref. 30: K. Takeuchi, et al., "Scaling effects predicted by WCOBRA/TRAC for UPI plant best estimate LOCA", Nuclear Engineering and Design, **186**, 257–278, (1998).

Ref. 31: Transient and accident analysis methods, U.S. NRC, Regulatory Guide 1.203, Dec. 2005.

Ref. 32: G.E. Wilson, B.E. Boyack, "The role of the PIRT process in experiments, code development and code applications associated with reactor safety analysis", Nuclear Engineering and Design, **186**, 23–37 (1998).

Ref. 33: T. Wickett (ed.), Report of the Uncertainty Methods Study for Advanced Best Estimate Thermal-Hydraulic Code Applications, Vols. I & II, OECD/CSNI Report, NEA/CSNI/R(97)35, 1997.

Ref. 34: W. Wulff, "Scaling of thermo-hydraulic systems", Nuclear Engineering & Design, **163**, 359-395 (1996).

Ref. 35: G. Yadigaroglu, M. Zeller, "Fluid-to-fluid scaling for a gravity- and flashing-driven natural circulation loop", Nuclear Engineering and Design, **151**, 49–64 (1994).

Ref. 36: M.Y. Young, et al., "Application of code scaling applicability and uncertainty methodology to the large break loss of coolant", Nuclear Engineering and Design, **186**, 39–52 (1998).

Ref. 37: B.J. Yun, et al., "Scaling for the ECC Bypass Phenomena during the LBLOCA Reflood Phase", Nuclear Engineering & Design, **231**, 315-325 (2004).

Ref. 38: N. Zuber, "Appendix D: a hierarchical, two-tiered scaling analysis, an integrated structure and scaling methodology for severe accident technical issue resolution. US NRC, Washington, DC 20555, NUREG/ CR-5809, Nov. 1991.

Ref. 39: N. Zuber, et al., "An integrated structure and scaling methodology for severe accident technical issue resolution: development of methodology", Nuclear Engineering and Design, **186**, 1–21 (1998).

Ref. 40: N. Zuber, "A General Method for Scaling and Analyzing Transport Processes", pp. 421-459, in M. Lehner, D. Mewes, U. Dinglreiter, R. Tauscher, *Applied Optical Measurements*, Springer, Berlin, 1999.

Ref. 41: N. Zuber, "The effects of complexity, of simplicity and of scaling in thermal-hydraulics", Nuclear Engineering and Design, **204**, 1–27 (2001).

Ref. 42: N. Zuber, et al., "Application of Fractional Scaling Analysis (FSA) to Loss of Coolant Accidents (LOCA). Part 1: Methodology Development", 11th Int. Top. Mtg. on Nuclear Reactor Thermal Hydraulics (NURETH-11), Avignon, France, Oct. 2-6, 2005.

7. NEW INITIATIVES: THE CFD4NRS SERIES OF WORKSHOPS, BENCHMARKING ACTIVITIES AND WEB PORTAL

7.1 The CFD4NRS Series of Workshops

The present Writing Group has provided evidence to show that CFD is a tried-and-tested technology and that the main commercial CFD vendors are taking active steps to quality-assure their software products by testing the codes against standard test data and through their participation in international benchmark exercises. However, it should always be remembered that the primary driving forces for the technology remain non-nuclear: aerospace, automotive, marine, turbo-machinery, chemical and process industries and, to a lesser extent, for environmental and biomedical studies. In the power-generation arena, we again find that the principal applications are non-nuclear: combustion dynamics for fossil-fuel burning, gas turbines, vanes for wind turbines, etc. Furthermore, the applications appear to focus mainly on design optimisation. This is perhaps not surprising since CFD can supply detailed information at the local level, building on a design originally conceived using traditional engineering approaches (though also computer-aided).

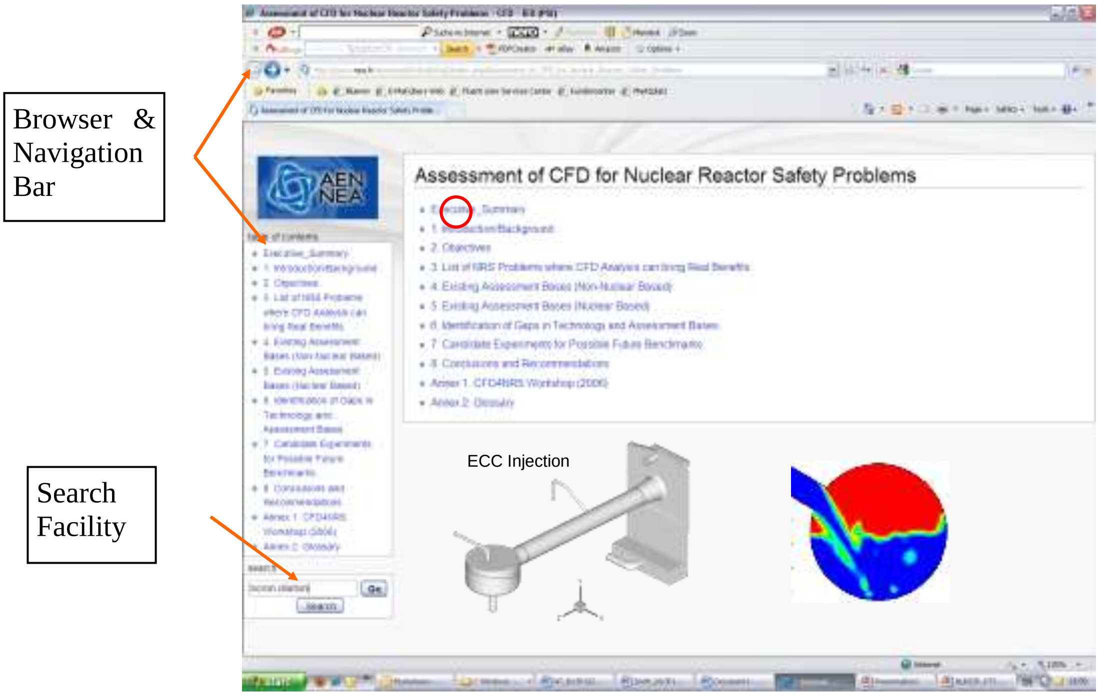
The most fruitful application of CFD in the nuclear power industry to date seems not to be a support to design, though this area is expected to increase in the near future, but rather to Nuclear Reactor Safety (NRS). The first step in fitting this particular application area into the "World of CFD", and as a direct product of the activities of the present Writing Group, was the organisation of the OECD/NEA and IAEA sponsored Workshop CFD4NRS Workshops, the first of which took place in Garching, Munich, Germany on 5-7 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, and were focussed in phenomenological areas such as: heat transfer; buoyancy; heterogeneous flows, natural circulation; free-surface flows; mixing in tee-junctions and complex geometries. Most papers related to topical NRS issues, such as: pressurized thermal shock; boron dilution, hydrogen distribution; induced breaks; thermal striping; etc. The use of Best Practice Guidelines (BPGs) was strongly encouraged. Selected papers appeared in a special issue of Nuclear Engineering and Design.

The second workshop in the series, XCFD4NRS, took place in Grenoble, France in September 2008. Here, the emphasis was more on new experimental techniques and two-phase CFD, addressing many of the NRS issues identified in Chapter 3 of this document. The workshop attracted 147 participants. There were 5 invited speakers, 3 keynote talks, 44 technical papers and 15 posters. Again, selected papers were collected in a special issue of the journal Nuclear Engineering and Design. The third workshop, CFD4NRS-3, was held in Washington DC in September 2010 and its proceedings appeared during 2011 with selected papers in a topical issue of Nuclear Engineering and Design in 2012. The fourth workshop, hosted by KAERI, took place in Daejeon, Rep. of Korea in September 2012 with the proceedings published in early 2014 (http://home.nea.fr/nsd/docs/2014/csni-r2014-4.pdf). The fifth workshop, CFD4NRS-5, was hosted by ETH Zurich in September 2014; at the time of writing, proceedings are being prepared and some papers have been selected for a special issue of Nuclear Engineering and Design.

The CFD4NRS workshops are a very useful addition to the more general conferences aimed at the nuclear technology community in that they are highly focused on CFD applications to nuclear safety issues and the special-effects validation experiments which qualify them. There is a strict review process for all papers. For the numerical analyses, the use of BPGs is now mandatory for acceptance, and the papers reporting experimental findings must contain data from local measurements that are suitable for CFD validation; the use of error bounds on the data are also strongly encouraged. Papers describing experiments which only provided data in terms of integral measurements (e.g., area-averaged data) were not accepted. The detailed programmes of the four workshops held to date are reproduced in Annex 1 of this report. Some background information, summary details, and recommendations made by participants are also included.

7.2 Moving the Writing Group Documents to the Web

The activities of the three OECD/NEA Writing Groups on CFD were concluded at the end of 2007 with the completion, or near completion, of their respective CSNI reports. It was recognized, like any state-of-the-art report, that these documents would only be up-to-date at the time of writing and, given the rapidly expanding use of CFD in the nuclear technology field, the information they contained would soon become outdated, though perhaps less so for the WG1 document dealing with BPGs. To preserve their topicality, improvements and extensions to the documents were foreseen, and for these to be made on a continuous basis. 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 was created on the NEA website using Wikimedia software. In a first step, the WG2 report, in the form in which it appeared in 2007 as an archival document, has been uploaded to provide on-line access. The WG1 document has since also been uploaded (though remains to be of restricted access), and the webpages for the WG3 document are currently under construction.



The current version of the main page for the WG2 webpage is shown above; a customized version is being prepared. There is unrestricted access to the webpages, which can be reached via the NEA website (www.oecd-nea.org) by following successively the links Work Areas: Nuclear Safety, CSNI, WGAMA. Listed are the main chapter headings of the WG2 document, the blue colour signifying that it is an active internal link to the detailed information. For example, clicking on the item Executive Summary (circled) opens up the pages containing the Executive Summary in its entirety as it exists in the original document. There is also an active scroll bar, and a hierarchical search facility for finding text strings in the pages. Navigation can be via the Navigation Bar or by use of the Browser functions.

The larger chapters are subdivided, and 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 being installed at this level too, to enable the user to navigate quickly to other parts of the document. The webpage addresses, for example to the commercial CFD sites, are also active, and it is planned to install a similar facility for the journal references too, 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 there, the Wiki software being 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, since this requires CSNI endorsement, but can communicate their suggestions to the website editors (the authors of this paper). In parallel, a beta version of the webpage will be maintained for installing updates prior to transfer to the static site. It will be the respective editor’s responsibility to review all new submissions, and implement them into the open-access version of the site. A special CFD Task Group has been set up within WGAMA (currently 38 members) to organize and coordinate the regular updating the websites. The changes made to the original WG2 document, as described in this revised version, will be uploaded to the website following CSNI approval.

7.3 CFD Benchmarking Exercises

7.3.1 Possible Benchmarks for Primary Circuits

Coolant mixing studies in primary/secondary circuits, e.g. thermal striping effects in or near a T-junction, and horizontal channel flows, were originally identified by the group as potential sources for future CFD benchmarking activities. Coolant mixing studies have been performed in the Rossendorf Coolant Mixing Model (ROCOM) test facility of FZD (now HZDR), the corresponding experiments being presented at the CFD4NRS Workshop by Kliem et al. (2006), and the CFD simulation results by Höhne and Kliem (2006). A paper on thermal mixing experiments in a T-junction was presented by Westin et al. (2006). In addition, Kliem (2007) and Vallée (2007) provided a detailed description of the test facilities at HZDR Rossendorf.

ROCOM

Kliem et al. (2007) give a detailed description of the ROCOM test facility, its measurement techniques and an error analysis of the experimental results. At the end of the report, the numerical simulation results for the steady-state and transient experiments with and without ECC injection were provided. The report is briefly summarised here.

The ROCOM test facility for the investigation of coolant mixing in the primary circuit of PWRs is described in detail in Chapter 5 of this document. Here, we just recall the principal features. The pressure vessel mock-up is made of Perspex, with detailed sub-models for the core barrel with the lower support plate and the core simulator, the perforated drum in the lower plenum and the inlet and outlet nozzles of the main coolant lines with diffuser elements. ROCOM is operated with de-mineralized water at ambient temperatures. Density differences, for instance for the simulation of boron dilution transients, are established by adding salt or ethyl alcohol.

Two loops of the test facility are equipped with fast-acting pneumatic gate valves. High-concentration salt slugs are generated between these valves. Measurement of the concentration fields is performed with high-resolution (in space and time) wire-mesh sensors that measure the electrical conductivity between two orthogonal electrode grids. In addition to the measurements in the cold legs, two further wire-mesh sensors with 4 radial and 64 azimuthal measuring positions in the downcomer and 193 conductivity measurements at the core entrance are installed. All sensors provide 200 measurements per second. Since a measuring frequency of 20 Hz is sufficient, ten successive images are averaged into one conductivity distribution. Experiments are repeated at least five times so as to quantify uncertainties due to time-dependent fluctuations of the flow field. The procedure for the error estimation is described in detail by Kliem et al. (2007).

The ROCOM experiments are very well suited for validation of CFD calculations, as they provide data with a high spatial and temporal resolution. The high quality of the data is consolidated by a thorough error analysis. Data for code validation comprise three mixing scenarios:

- Steady-state flow scenarios examining fluctuations in the boron concentration caused by sub-cooled water arriving from the steam generators;
- Transient flow scenarios including one or more operating loops, such as:
  - start-up of the main coolant pumps with a de-borated slug;
  - onset of natural convection occurring during a loss of coolant accident;
- Gravity-driven flows caused by large density gradients which can occur during ECC water injection.

The CFD4NRS paper of Kliem et al. (2006) gives an overview of these experiments. Data were made available from selected tests to form the basis of a benchmark activity within the 5th FWP FLOWMIX-R, but much more information is available on: (1) stationary experiments, in which the pumps in all loops are driven with a constant mass flow rate; (2) transient experiments, in which the start-up of a main coolant pump is simulated with a tracer (passive scalar) in one loop; and (3) experiments with density differences, to explore the effects of buoyancy-driven mixing for some low-flow cases.

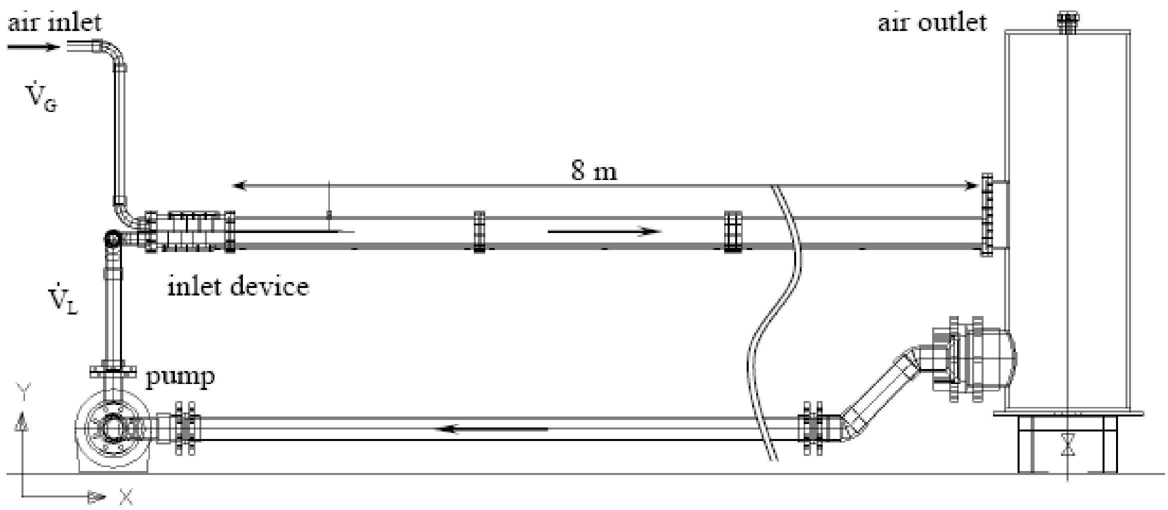
A number of the ROCOM experiments have already been simulated by different organisations, using a variety of CFD codes. Details are given in the table below.

The series of ROCOM experiments represents a solid data base of validation data for CFD simulation of the boron dilution event, and generally for in-vessel mixing phenomena. Due to lack of time and/or funding, the full potential of validation data remains largely unexplored. Benchmark exercises based on data from these experiments fulfil all the requirements of an NRS assessment matrix.

Experiments	Boundary conditions	Code	Organization
Stationary experiments	4 loop operation at nominal flow (equal flow rates)	CFX-4 CFX-5 CFX-5 CFX-10 FLUENT FLUENT Trio_U	FZD FZD GRS Uni Pisa VUJE AEKI CEA & Uni Pisa
	4 loop operation at reduced flow (equal loop flow rates)	CFX-10	Uni Pisa
	4 loop operation (different flow rates)	CFX-4 FLUENT	FZD AEKI
	3 loop operation (equal flow rate)	FLUENT	VUJE
Transient experiments	Start-up of the pump in loop 1 up to nominal flow rate (different slug sizes)	CFX-4 CFX-5 FLUENT CFX-5 CFX-10	FZD FZD FORTUM NRG Uni Pisa
	Start-up of the pump in loop 1 up to reduced flow rate	CFX-5	NRG
	Start-up of the pump in loop 1 up to nominal flow rate (velocity measurements)	CFX-10	FZD
Experiments on ECC-water injection	$\Delta\rho/\rho$ = 10%, Flow rate=5 %	CFX-5 TRIO-U CFX-5	FZD CEA GRS
	$\Delta\rho/\rho$ = 5 %, Flow rate=5 %	CFX-5	NRG

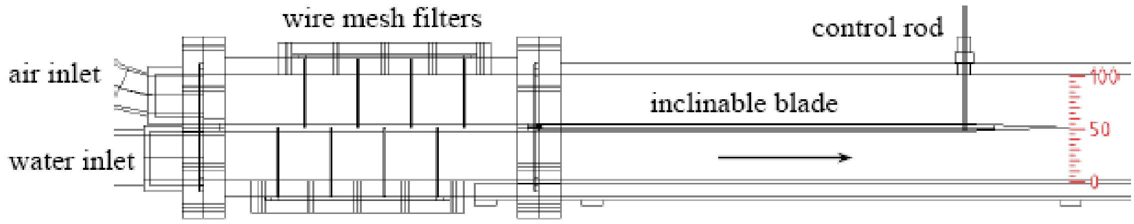
HAWAC SEPARATED FLOW BENCHMARK

In different scenarios of Small-Break Loss-of-Coolant Accidents (SB-LOCAs), stratified two-phase flow regimes can occur in the main cooling lines of PWRs. The corresponding horizontal air-water flows have been investigated in the Horizontal Air/Water Channel (HAWAC) of HZDR on behalf of the German Federal Ministry of Economy and Technology.



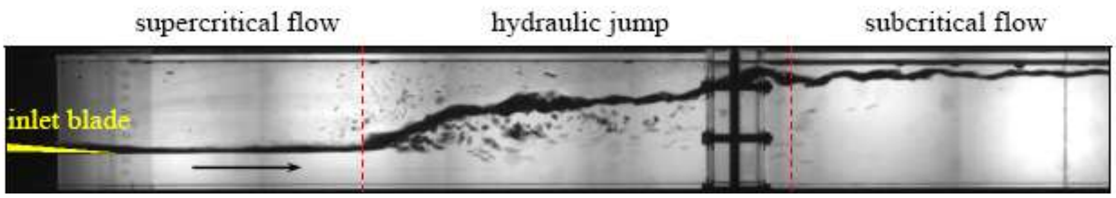
The HAWAC facility, shown in schematic form in the Figure above, provides observations of co-current slug flow. A special inlet device provides well-defined inlet boundary conditions via a separate injection of water and air into the test section. The test section is 8 m long and of cross-section is 100×30 mm; this gives a length-to-height ratio of 80.

The inlet device (Figure below) is designed for separate injection of water and air into the channel: the air flows through the upper part and the water through the lower part of the device. In order to mitigate flow perturbations at the inlet, 4 wire-mesh filters are mounted in each part of the inlet device, providing homogenous velocity profiles at the test section inlet. Moreover, the filters produce a pressure drop that attenuates the effect of the pressure surge created by slug flow on the fluid supply systems.



Air and water come in contact at the edge of a 500 mm long blade, which divides the two phases downstream of the filter segment. The inlet cross-section for each phase can be controlled by inclining this blade. Use of the filters and the blade provides well-defined inlet boundary conditions for the associated CFD simulations.

If the velocities at the end of the blade are similar, air and water merge smoothly together, otherwise a perturbation can be introduced in the channel. At high water flow rates, especially when the inlet blade is inclined downwards, a hydraulic jump can be formed in the test-section. The hydraulic jump is the turbulent transition zone between supercritical and subcritical flows.



In the supercritical region, the flow is always stratified, whereas after the hydraulic jump (i.e. in the subcritical region) typical two-phase flow regimes are observed (e.g. elongated bubble flow and slug flow). The position of the hydraulic jump in the channel depends on the flow rates and the inlet blade inclination. When a hydraulic jump occurs, its position strongly influences the inlet length needed for the generation of slug flow. A flow pattern map was generated on the basis of visual observations of the flow structure at different combinations of gas and liquid superficial velocities. The observed flow patterns were stratified flow, wavy flow, elongated bubbly flow and slug flow.

Sub-categories were defined to consider the slug generation frequency and the appearance of elongated bubbles in the channel: sporadic (transition regime), periodic, but only one type of structure (either slug or elongated bubble), and periodic with several types of structures present simultaneously.

Due to the rectangular cross-section, the flow can be observed very well from the side of the duct. To make quantitative observations, the flow was filmed with a high-speed video camera at 400 frames per second. The single pictures are stored in bitmap format and depict, for example, the generation of slugs.

The water level in a cross-section as a function of time was also measured, with a frequency of 400 Hz, which corresponds to the frame rate of the high-speed camera. Since direct comparison of the measured water levels against CFD predictions is difficult, a statistical approach is proposed. First, a time-averaged water level is calculated and bounded by the standard deviation in each cross-section. This results in a mean water level profile along the channel which reflects the structure of the interface. Further, the standard deviation σ quantifies the spread of the measured values which originate in the dynamics of the free surface. Another possibility is to plot the probability distribution of the water levels.

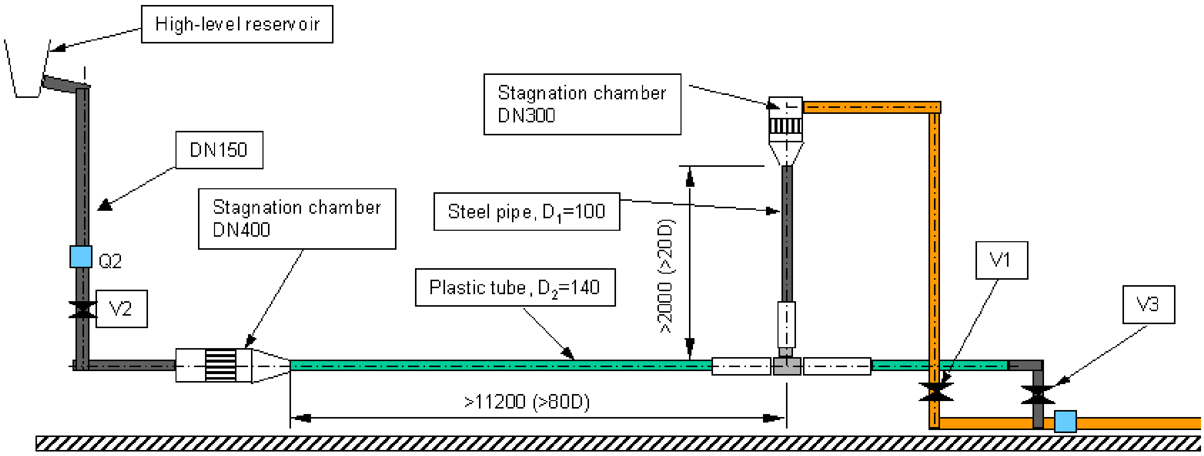
The picture sequence recorded during slug flow was compared with CFD simulation results obtained using ANSYS-CFX-10, the mesh consisting of 600,000 control volumes. Turbulence was modelled separately for each phase using the k-ω based Shear Stress Transport (SST) model. Results showed that with an Euler-Euler model approach, behaviour of slug generation and propagation seen in the experiment could be qualitatively reproduced, but quantitative comparisons indicate that further model improvement is needed. Again, data are available of sufficiently high quality to validate the treatment of separated flows in CFD codes (without mass exchange between the phases).

VATTENFALL T-JUNCTION FACILITY

Unsteady temperature fluctuations in duct systems can lead to thermal fatigue in duct walls; examples exist from nuclear power plants in which thermal fatigue has been the cause of leaks in the primary and secondary circuits. A possibility to mitigate the risk is to install devices to enhance mixing. Static mixers have, for example, been developed at Vattenfall R&D since the early 1980s, and are installed in some Swedish nuclear power plants. The problem is that such devices are expensive, and increase pressure drops. Therefore, significant cost reduction can be achieved by accurately predicting conditions which promote thermal fatigue, and then adjusting operational conditions accordingly. This is a fertile area for CFD simulation.

Analysis of crack growth due to cyclic loading requires accurate description of both the amplitudes and the frequencies of the thermal fluctuations near pipe walls. Standard CFD approaches based on RANS cannot provide data of this type, and careful validation of advanced turbulence models (e.g. DES or LES) needs to be carried out. This requires appropriate experimental data measurements. Such tests have been carried out at the Älvkarleby Laboratory of Vattenfall R&D.

The test rig was designed to simulate a typical T-junction in a nuclear power plant, using a model scale of 1:1.5. The horizontal cold water main pipe had a diameter of 190 mm in the model tests, and the water temperature was approximately 25°C. The vertical hot water branch pipe was connected from below to the main pipe, and had a diameter of 123 mm and a water temperature of approximately 60°C. The pipes were made of acrylic glass to allow optical access. The experimental set-up also included upstream bends in order to obtain realistic flow conditions approaching the T-junction. An outline of the model geometry is shown here, based on a simulation using the FLUENT code.



In addition to the temperature measurements, flow visualizations were used. Also, a limited number of velocity measurements were carried out using Pitot-tube and Laser Doppler Velocimetry (LDV). However, the quality of the velocity measurements is not considered trustworthy enough for CFD validation.

A number of different ratios between the cold (Q2) and hot (Q1) water flows were tested. Calculations have been carried out for three of the test cases, and the test conditions are summarized in Table 1. The penetration of the hot branch flow into the main pipe is significantly different between Tests 9 and 11, which are illustrated in the flow visualizations in Figure 3. The mixing is characterized by large-scale fluctuations, which is more evident in the cases with smaller flow ratios (Test 10 and 11).

Table I: Test conditions in the simulations. (*) In the simulation of test 10 a constant viscosity was used which gave a slightly different Reynolds number in the hot water pipe.

Parameter	**Test 9**	**Test 10**	**Test 11**
Q_1 (l/s)	20.0	20.0	20.0
Q_2 (l/s)	112.5	56.3	47.8
Q_2/Q_1	5.6	2.8	2.4
T_1 (°C)	65.9	59.8	59.9
T_2 (°C)	27.3	24.0	25.7
Re_1	4.7×10^5	3.2×10^5	4.3×10^5
Re_2	8.8×10^5	5.8×10^5	3.6×10^5

CFD results obtained from RANS and URANS simulations showed very poor comparisons, indicating that scale-resolving methods such as LES and DES are essential for such applications. Several different models have been used in the calculations and the Table below summarizes some of the numerical settings and material properties used in the LES and DES calculations.

Settings	Test 9	Test 10	Test 11
FLUENT version	6.2.16	6.1.22	6.2.5
Model	DES and LES	LES	DES
DES model	Spalart-Allmaras	-	Spalart-Allmaras
SGS model (LES)	Dyn. Smagorinsky	Smagorinsky	-
Momentum	Bounded central differences (BCD)	Central diff. 2nd order Upwind, QUICK	BCD
Pressure	2nd order	Standard	Presto
Energy	QUICK	QUICK	QUICK
Pressure-velocity coupling	Fractional step	SIMPLE	PISO and SIMPLE
Gradient option	Node based	Cell based	Cell based
Transient scheme	NITA	ITA	ITA
Time step	1 ms and 0.25 ms	0.5 ms (and 2ms)	2 ms and 1 ms
Iterations/time step	-	-	15 and 30
Density	Curve fit	Boussinesq	Boussinesq and curve fit
Dynamic viscosity	Curve fit	6.58×10^{-7} (const.)	Curve fit
C_p	4178.6 (const.)	4178.6 (const.)	4182.5 (const.)
Thermal Conductivity	0.6306 (const.)	0.6306 (const.)	0.62 (const.)

Ref. 1: Baker, O. (1954), Simultaneous Flow in Oil and Gas,Oil and Gas J., 53, 185- 195, 1954

Ref. 2: Braillard, O., Jarny, Y. and Balmigere, G. (2005) Thermal load determination in the mixing Tee impacted by a turbulent flow generated by two fluids at large gap of temperature, ICONE13-50361, 13th International Conference on Nuclear Engineering, Beijing, China, May 16-20, 2005

Ref. 3: Cartland Glover, G. M.; Höhne, T.; Kliem, S.; Rohde, U.; Weiss, F.-P.; Prasser, H.-M. (2007), Hydrodynamic phenomena in the downcomer during flow rate transients in the primary circuit of a PWR, Nucl. Eng. Design, vol. 237, pp. 732-748

Ref. 4: Harleman, M. (2004) Time dependent computations of turbulent thermal mixing in a T-junction. Report FT-2004-685, Forsmarks Kraftgrupp AB

Ref. 5: Hemström, B., et al. (2005) Validation of CFD codes based on mixing experiments (Final report on WP4) EU/FP5 FLOMIX-R report, FLOMIX-R-D11. Vattenfall Utveckling (Sweden)

Ref. 6: Höhne, T., Kliem, S. (2006), “Coolant mixing studies of natural circulation flows at the ROCOM test facility using ANSYS ANSYS-CFX”, CFD4NRS 2006, 05.-07.09.2006, Garching, Germany, Proceedings, Paper 23

Ref. 7: Janobi, M. (2003) CFD calculation of flow and thermal mixing in a T-junction (steady state calculation), Report U 03:69, Vattenfall Utveckling AB

Ref. 8: Jungstedt, J., Andersson, M. and Henriksson, M. (2002) Termisk blandning i T-stycke – Resultatrapport. Report U 02:134, Vattenfall Utveckling AB, 2002

Ref. 9: Kliem, S., Rohde, U., Sühnel, T., Höhne, T., Weiss, F.-P. (2007), „ A test facility for the investigation of coolant mixing inside the reactor pressure vessel of PWRs“, Draft report, personnel communication.

Ref. 10: Kliem, S., Sühnel, T., Rohde, U., Höhne, T., Prasser, H.-M., Weiss, F.-P. (2006), „ Experiments at the mixing test facility ROCOM for benchmarking of CFD-codes“, CFD4NRS 2006, 05.-07.09.2006, Garching, Germany, Proceedings, Paper 17.

Ref. 11: Lycklama à. Nijeholt, Jan-Aiso; Höhne, T. (2006), On the application of CFD modeling for the prediction of the degree of mixing in a PWR during a boron dilution transient, ICAPP ‘06, ANS, 04.-08.06.2006, Reno, NV, USA, Proceedings, Paper 6155

Ref. 12: Mandhane, J. M., Gregory, G. A. and Aziz, K., (1974), A Flow Pattern Map for Gas-Liquid Flow in Horizontal Pipes: Predictive Models, Int. J. Multiphase Flow, 1, 537-553, 1974

Ref. 13: Ohtsuka, M., Kawamura, T, Fukuda, T., Moriya, S., Shiina, K., Kurosaki, M., Minami, Y. and Madarame, H. (2003) LES analysis of fluid temperature fluctuations in a mixing Tee pipe with the same diameters, ICONE 11-36064, 11th International Conference on Nuclear Engineering, Tokyo, Japan, April 20-23, 2003

Ref. 14: Péniguel, C., Sakiz, M., Benhamadouche, S., Stephan, J.-M. and Vindeirinho, C. (2003) Presentation of a numerical 3D approach to tackle thermal striping in a PWR nuclear T-junction, PVP/DA007, Proceedings of ASME PVP, July 20-24, 2003, Cleveland, USA

Ref. 15: Prasser, H.-M., Böttger, A., Zschau, J. (1998), A new electrode-mesh tomograph for gas-liquid flows, Flow Measurement and Instrumentation 9, 111-119

Ref. 16: Prasser, H.-M, Grunwald, G., Höhne, T., Kliem, S., Rohde, U., Weiss, F.-P. (2003), Coolant mixing in a PWR - deboration transients, steam line breaks and emergency core cooling injection - experiments and analyses, Nuclear Technology, vol. 143 (1), pp. 37-56

Ref. 17: Rohde, U., Kliem, S., Höhne, T., Karlsson, R. et al. (2005), Fluid mixing and flow distribution in the reactor circuit: Measurement data base, Nucl. Eng. Design, vol. 235, pp. 421-443

Ref. 18: Vallée C. (2007), “Stratified two-phase flow experiments in the horizontal air/water channel (HAWAC)” FZD-report, personnel communication

Ref. 19: Vallée, C., Höhne, T., Prasser, H.-M. Sühnel T. (2006), Experimental investigation and CFD simulation of horizontal air/water slug flow, Kerntechnik, Vol. 71 (3), 95-103

Ref. 20: Veber, P. and Andersson, L. (2004) CFD calculation of flow and thermal mixing in a T-junction – time dependent calculation. Teknisk not 2004/7 Rev 0. Onsala Ingenjörsbyrå AB

Ref. 21: Veber, P. and Andersson, L. (2004) CFD calculation of flow and thermal mixing in a T-junction – time dependent calculation – Part 2. Teknisk not 2004/21 Rev 0. Onsala Ingenjörsbyrå AB

Ref. 22: Westin, J. (2005) Thermal mixing in a T-junction: Steady and unsteady calculations, Report U 05:118, Vattenfall Utveckling AB

Ref. 23: Westin, J., Alavyoon, f., Andersson, L., Veber, P., Henriksson, M., Andersson, C., (2006), “Experiments and unsteady CFD-calculations of thermal mixing in a T-junction”, CFD4NRS 2006, 05.-07.09.2006, Garching, Germany, Proceedings, Paper 25

7.3.2 Possible Containment Benchmarks

Experiments relevant to (primarily single-phase) containment issues involve considerations such as thermal hydraulics, hydrogen distribution and hydrogen combustion. Though many experiments have been performed over the last twenty years (some of which being the object of international standard problem exercises), most have been dedicated to the validation of lumped-parameter containment codes. Data suitable for CFD validation have only appeared over the last ten years with the construction of new experimental facilities allowing better control of initial and boundary conditions, and the use of state-of-the-art instrumentation techniques for detailed measurements.

A review of data suitable for validating CFD codes for containment issues was performed in part in the framework of the ECORA project (Scheuerer et al., 2005). Also, the OECD/NEA are supporting ongoing tasks leading to the elaboration of a so-called Containment Code Validation Matrix, which addresses both lumped-parameter and CFD codes. As well as the distinction between containment thermal-hydraulics and hydrogen combustion tests, one should also distinguish between so-called Separate Effect

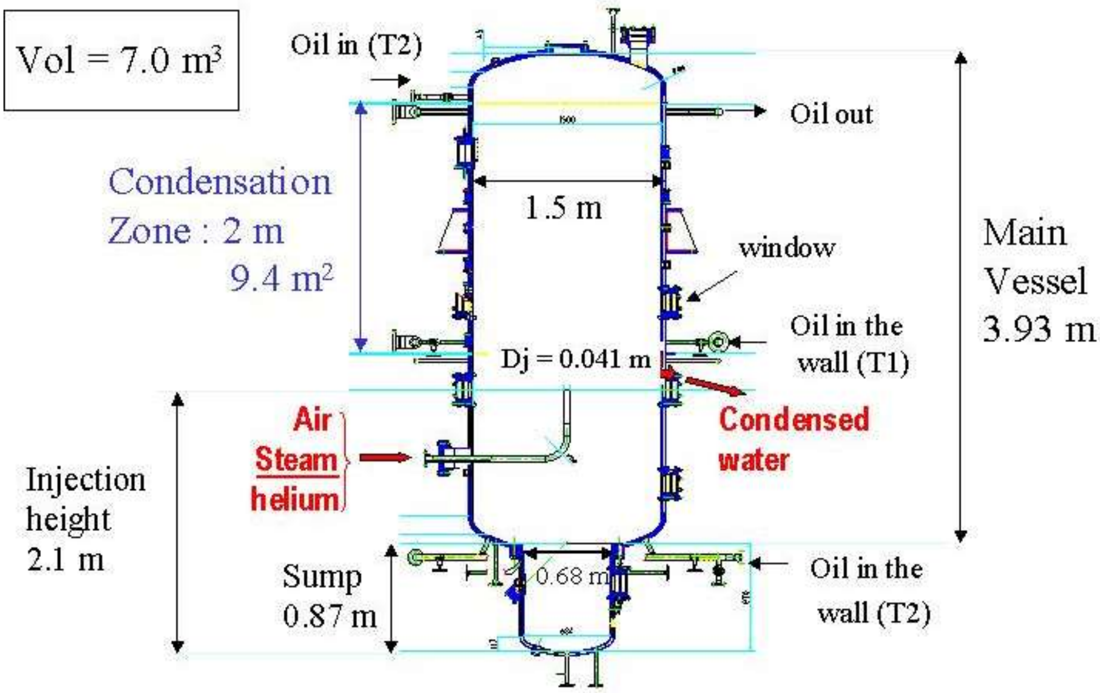
Test facilities and Coupled Effect Test facilities – this distinction being quite often associated with the size of the facility.

A validation test matrix may already be defined, based on experiments performed using small-scale and large-scale facilities, and code comparisons are currently underway using data from such large-scale facilities as HDR (Mueller-Dietsche and Katzenmeier, 1985, Scholl, 1983), PANDA (Yadigaroglu and Dreier, 1998; Paladino et al., 2007; Andreani et al., 2007) and RUT (Breitung et al, 2005, Studer and Galon, 1997), as well as some newly dedicated ones, such as MISTRA (Caron-Charles, 2002), TOSQAN (Brun et al, 2002, Kljenak, 2006) and ENACCEF (Bentaïb, 2005). The Table below gives a summary of ongoing activities.

	Facilities/tests	**Initial mixture**	**Phenomena**
DISTRIBUTION	AECL LSGMF	Air-helium	Jet, stratification, turbulence
	Phebus FPT0	Air-steam-hydrogen	Jet, condensation (in presence of H2)
	Phebus FPT1	Air-steam- hydrogen	Jet, condensation (in presence of H2)
	MISTRA helium tests	Air-helium	Jet, stratification, turbulence
	MISTRA ISP47	Air-steam-helium	Jet, condensation (in presence of He), stratification
	MISTRA MICOCO	Air-steam	Buoyant plume, condensation
	MISTRA M1	Air-steam	Jet, condensation
	MISTRA M2	Air-steam	Jet, condensation
	MISTRA M3	Air-steam	Jet, condensation, 3D flow
	TOSQAN 1	Air-steam	
	TOSQAN 2	Air-steam	
	TOSQAN 3	Air-steam	
	TOSQAN 6	Air-steam	
	TOSQAN 7	Air-steam	
	TOSQAN 8	Air-steam	
	TOSQAN 9b	Air-steam	
	TOSQAN ISP47	Air-steam-helium	Jet, condensation (in presence of He)
	MAEVA mock-up	Air-steam	Jet release, condensation, concrete structure heat-up
	PANDA SETH tests	Air, air-steam, steam	Horizontal jets, vertical plumes, near-field velocity distribution, stratification, condensation, gas (helium or steam) transport in a multi-compartment geometry
	PANDA SETH test 17	Air	Horizontal buoyant jet
	PANDA SETH test 9	Air	Near-wall plume
SPRAY	TOSQAN 101	Air-steam	Condensation by spray
RECOMBINER	KALI-H2, test 008	Air-steam-hydrogen	Recombination by PAR
COMBUSTION	Driver MC012	Air-hydrogen	H2 combustion
	RUT HYC01	Air-hydrogen	H2 combustion
	RUT Sth064	Air-hydrogen-steam	H2 combustion
	RUT STM4	Air-steam-hydrogen	H2 detonation
	HDR E12.3.2	Air-hydrogen	H2 deflagration
	BMC ex29	Air-hydrogen	H2 deflagration
	ENACCEF – test1	Air-hydrogen	H2 deflagration

TOSQAN FACILITY

The TOSQAN experiment (see Figure) is a closed cylindrical vessel (7 m3, i.d. 1.5 m, total height of 4.8 m, condensing height of 2 m) into which steam or non-condensable gases are injected through a vertical pipe located on the vessel axis. This vessel has thermostatically controlled walls so that steam condensation may occur on one part of the wall (the condensing wall, CW), the other part being superheated (the non-condensing wall, NCW). Over 150 thermocouples are located in the vessel (in the main flow and near the walls). 54 sampling points for mass spectrometry are used for steam volume fraction measurements. Optical accesses are provided by 14 overpressure resistant viewing windows permitting non-intrusive optical measurements along an enclosure diameter at 4 different levels (LDV and PIV for the gas velocities, Raman spectrometry for steam volume fractions.

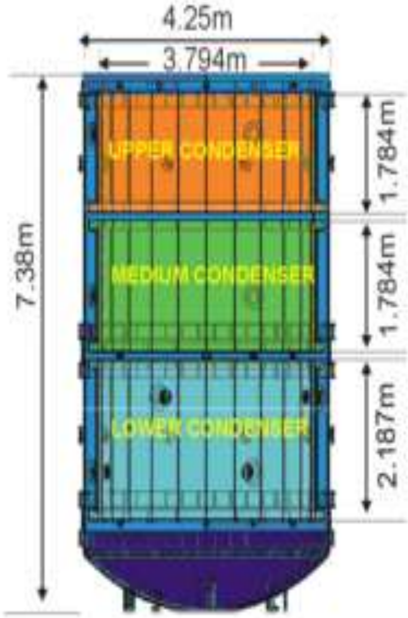


The condensation tests in TOSQAN consist of steam injection into the enclosure, initially filled with air at atmospheric pressure, the NCW and the CW having already reached their nominal temperatures. After a transient stage corresponding to enclosure pressurization, a steady-state is reached in which the steam injection and the condensation flow rates are equal. This corresponds to constant enclosure total pressure and thermal equilibrium.

Qualification of TONUS (Bentaib, 2006) is performed on two levels: a global level on which only the mean pressure during steady-state is evaluated, and a local level for which comparison of gas temperature, steam concentration and velocity profiles at different locations are given. CFD simulations have been carried out using the TONUS-CFD code (the lumped-parameter version of the code was also used). Total pressure is predicted satisfactorily, and local gas temperatures are also well reproduced, as are gas temperature horizontal profiles below the injection point. Similar curves can be obtained for all the TOSQAN tests. The code-experiment temperature difference is generally around 1-3°C.

MISTRA FACILITY

The MISTRA facility is a stainless steel cylindrical containment of volume 100 m^3. The internal diameter of 4.25 m and height of 7.3 m were chosen to correspond to a linear length scale ratio of 1:10 with a typical French PWR containment. The vessel comprises 2 cylindrical shells, flanged together, and flat top and bottom sections, also flanged. The vessel itself is not temperature-regulated, but thermally insulated with 20 cm of rock wool. Prior to the experiments; the facility is usually preheated by steam injection (pre-heating phase).



Three cylindrical condensers are inserted inside the containment (see Figure above), close to the vessel walls. The external parts of the condensers are insulated with synthetic foam and viewing windows are installed for laser measurements. Gutters are installed to collect and quantify the condensates. A diffusion cone including a porous medium is designed for gas injection and steam/gas (helium simulating hydrogen or other gases) mixing. The injection velocity profiles are flat. Injection gas flow rates are controlled and measured with sonic nozzles that ensure a constant value independently of the downward operating conditions. The different gases can be heated up to 220°C, which is the design temperature of the facility.

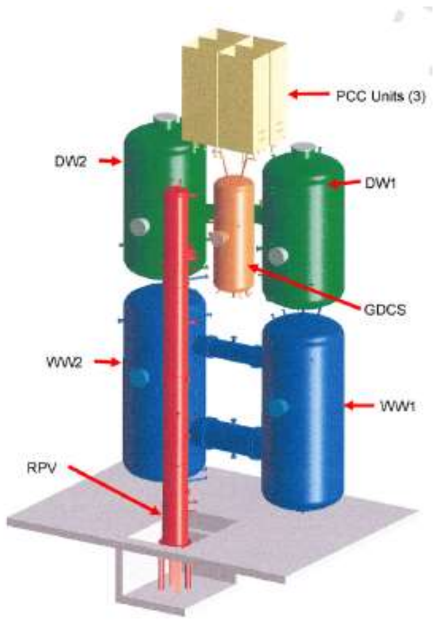
The measurements performed in MISTRA are related to pressure, temperature (gas and wall), gas composition (steam, air, helium), velocity and condensed mass flow rate. They are all simultaneously and continuously recorded over the whole test period, except for gas concentration measurement, which is performed using sampling. Laser Doppler Velocimetry or Particle Image Velocimetry is employed to measure instantaneous velocity profiles and turbulence characteristics. The TONUS validation procedure for the MISTRA tests follows that of TOSQAN, in which a two-level validation procedure is employed: a global level, on which only the mean pressure during steady state is evaluated, and a local level, for which comparison of gas temperature, steam concentration and velocity profiles at different locations are given. Overall, code-experiment comparisons are good, for both global values, such as total pressure, and local gas temperature, velocity value and concentrations. Data from these tests have been assembled within the ISP 47 benchmark.

Recent tests in MISTRA have focussed on flows within a compartmented geometry (see figure 12), in which obstacles prevent the condensation-induced natural convection movements, thereby creating conditions favourable to thermal and mass concentration gradients.

However, it should be mentioned that most of the validation so far has dealt with steady state flows, so that the focus of future tests and validation will be on transient flows with thermal and gas stratification and break-up.

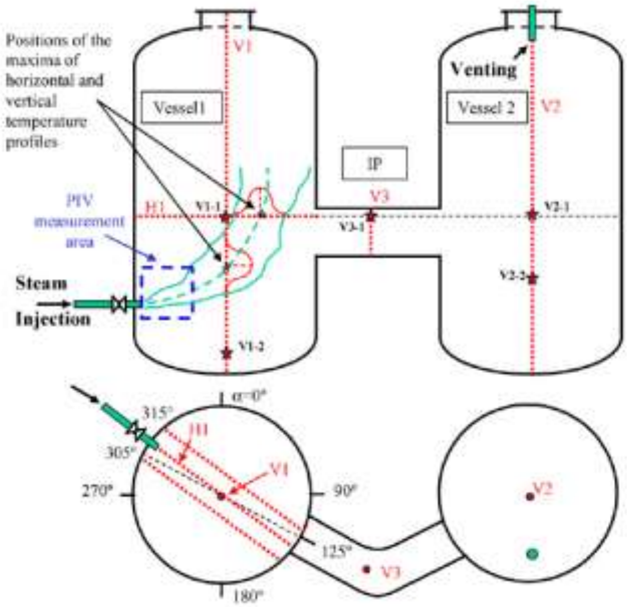
PANDA FACILITY

PANDA is a large-scale thermal-hydraulics test facility designed and used for investigating containment system behaviour and phenomena for different Advanced Light Water Reactor designs and large-scale separate-effect tests (Yadigaroglu and Dreier, 1998). The facility consists of large interconnected vessels, condensers and open water pools (see Figure below). Its modular structure provides flexibility for investigating a variety of different integral and local phenomena. The total height of the facility is 25m and is designed for 1MPa and 200^{o}C maximum operating conditions. Auxiliary systems are available to add or remove water, steam or gas to any vessel at desired conditions (temperature, pressure).



Though originally conceived to test the concept of passive decay heat removal from the containment of the Simplified Boiling Water Reactor of General Electric in the US (at 1/25th volumetric scale, but 1:1 in height), it was reconfigured for the European version of the Simplified Boiling Water Reactor, the ESBWR, and, in the BC series, building condensers were added to examine the containment cooling concept put forward for the SWR-1000, an alternative passive Boiling Water Reactor design proposed by Siemens. More recently (Auban et al., 2007), the two Dry-Well tanks have been used to perform special-effect tests in the OECD/SETH test series, in which jets/plumes, gas mixing and stratification have been investigated. Each of the two Dry-Well (DW) vessels is of height 8m, diameter 4m and an inner volume of 90m^3, connected by a large (～1m) diameter interconnecting pipe (IP), and have been heavily instrumented for these tests. In addition, the vessels and adjacent piping are covered with a 200 mm-thick layer of insulation rock-wool to minimize heat losses (estimated at 9 kW for an operating temperature of 110^{o}C).

The instrumentation consists of numerous sensors for the measurements of fluid and wall temperatures, absolute and differential pressures, flow rates, valve states and heater power. The facility is also equipped with a gas concentration measurement system utilising a mass spectrometer. A ‘Particle Image Velocimetry’ (PIV) system has been set-up for measuring 2D fluid velocity fields in some selected areas.



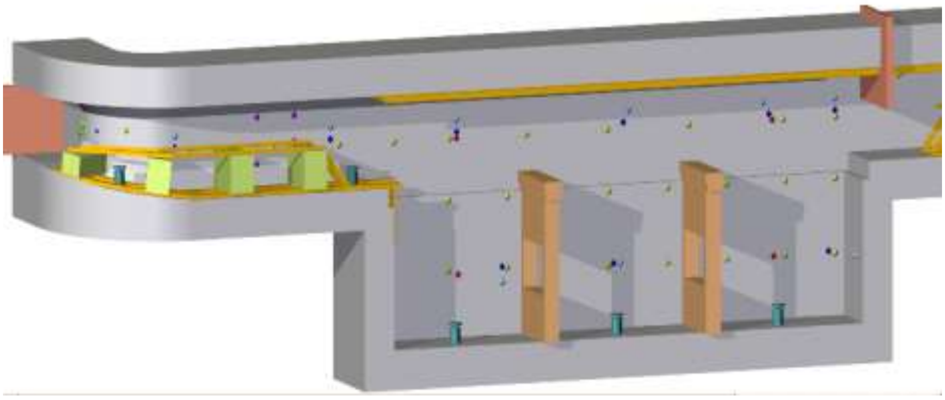
The Figure shows in schematic form the layout for one of the early first tests in the series. Steam is injected horizontally into DW1, which is initially filled with air. Flow rates are adjusted to reproduce wall plumes, free plumes (illustrated in the Figure) and jet-like behaviour, as appropriate. Venting takes place at the top of the second vessel. The well-characterized initial and boundary conditions of these tests are in accordance with the objectives of the experimental campaign, and provide suitable data for CFD validation. Moreover, the test results have been confirmed by repetitions of each test.

The dense instrumentation grid provides the time history of temperature and gas composition during the transient enabling the flow structure in the vessels and the stratification patterns in them to be determined. Data from the tests will come into the public domain during 2009.

RUT FACILITY

The RUT facility is operated by Kurchatov Institute, and the experimental tests here reported here were carried out in this facility in the frame of the HYCOM (Breitung, 2005) project. A schematic of the RUT facility is shown in the Figure.

The facility can be described as a large duct with variable cross-section, and subdivided into a number of compartments. A channel (35 m long, and of volume 180 m^3) with obstacles is connected to a block of 3 compartments (≈60 m^3 each, divided by walls with Blockage Ratio (BR) equal to 0.3) and then to another channel (≈60 m^3). The gas distribution system provided the possibility to arrange different hydrogen concentrations in the two parts of the facility. Local H_2 concentrations were measured with a sampling method using eight sampling ports with an accuracy of 0.25 % vol. The mixture was ignited with a weak electric spark. The measurement system included 45 collimated photodiodes to measure local flame arrival times, 16 piezoelectric pressure transducers (0.5 Hz ÷ 100 kHz) and 16 piezoresistive pressure transducers (0 ÷ 1 kHz), and 10 integrating heat-flux meters (0.02 ÷ 10 Hz.



Large-scale tests carried out in the RUT facility were aimed at studying the processes of turbulent flame propagation in multi-compartment geometries, and in non-uniform mixtures on typical reactor length scales. Tests HYC01 and STH6 (see Table) are chosen here to illustrate the ability of TONUS to simulate slow and fast deflagration regimes.

Test case	*Initial H_2 molar fraction*	*Initial air molar fraction*	*Initial H_2O molar fraction*	*P_i (Pa)*	*T_i (K)*	*Regime*
HYC01	0.1	0.9	0.	100200	290.7	Slow deflagration
STH06	0.162	0.388	0.45	100150	373	Fast deflagration

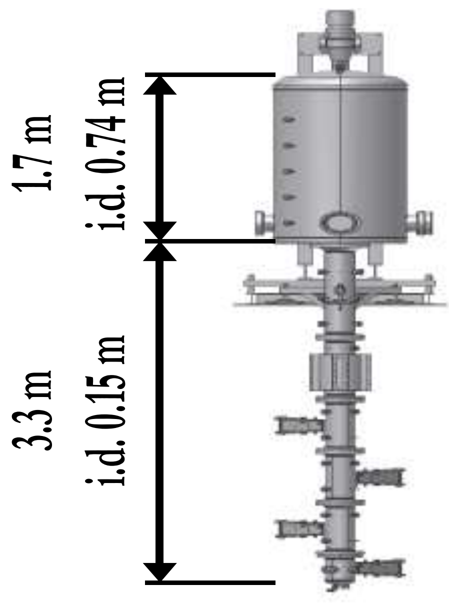
The TONUS model correctly calculates the slope of the pressure rise and maximum overpressure. For fast deflagrations, the model shows relatively little sensitivity to the grid size, except for the peaks that were captured better with the finer mesh.

Though previously used already for a benchmarking exercise, experiments from the TONUS and MISTRA series continue to provide valuable data for CFD validation. It should be recalled that not all tests involve two-phase aspects.

ENACCEF FACILITY

The ENACCEF facility is operated by CNRS, France in the frame of a cooperation agreement with IRSN. A sketch of the test section, including dimensions is given in the attached Figure. The facility is designed for the study of hydrogen flame propagation, and is a combination of two parts. The acceleration tube (3.2 m long and 154 mm i.d.), is mounted at the lower end, and at its lowest point is equipped with two tungsten electrodes to initiate a low energy ignition. At a distance of 1.9 m from the ignition point, three rectangular quartz windows (40 mmx300 mm optical path) are mounted flush with the inner surface, two of them are opposed to each other, while the third is perpendicular to these. The windows allow the recording of the flame front during its propagation along the tube using either a shadowgraph or a tomography system. The tube is also equipped with 11 small quartz windows (optical diameter: 8 mm, thickness: 3 mm) distributed along its length. UV-sensitive photomultiplier tubes (HAMAMATSU, 1P28) are placed in front of these windows in order to detect the flame passage. Several high speed pressure transducers, (7 from CHIMIE METAL and 1 PCB) are mounted flush with the inner surface of the tube in order to monitor the pressure variation in the tube as the flame propagates.

The dome (1.7 m long, 738 i.d.) is connected to the upper part of the acceleration tube via a flange in which a diaphragm can be mounted in order to isolate the two parts when needed. This part of the setup is also equipped with three silica windows (optical path: 170 mm, thickness: 40 mm), perpendicular to each other, two by two. Through these windows, the arrival of the flame can be recorded via a schlieren or a tomography system. Five UV-sensitive photomultiplier tubes, of the same series as above, are mounted across the silica windows (optical diameter: 8 mm, thickness: 3 mm) in order to detect the flame as it propagates through the dome. The pressure build up in this part is monitored via a PCB pressure transducer mounted at the ceiling of the dome.



Several obstacles can be inserted in the acceleration tube. Two different shapes have been used, annular obstacles of different blockage ratios (from 0.33 up to 0.63) and hexagonal mesh grids (with holes of 10 mm diameter spaced by 15 mm) of blockage ratio 0.6.

The ENACCEF test matrix includes homogenous tests and heterogonous tests with hydrogen gradient concentrations present in some tests. A version of the TONUS CREBCOM code has been validated against flame speed propagation tests in this series. Code performance was generally satisfactory, but points of discrepancy remain, thought to be due to the influence of turbulence on combustion speed and heat loss effects, which were not taken into account in the model.

Tests in the ENACCEF series have been carefully performed, and the data collected are of high quality. There is good potential here for benchmarking activities for other containment codes.

Ref. 1: Andreani, M., Haller, K., Heitsch, M., Hemström, B., Karppinen, I.,Macek, J., Schmid, J., Paillere, H., Toth, I. (2007), "A Benchmark Exercise on the use of CFD Codes for Containment Issues using Best Practice Guidelines: a Computational Challenge", *Nuclear Eng. Design* (in press), Ref: Nuclear Eng. Design (2007), doi:10.1016/j.nucengdes.2007.01.021.

Ref. 2: Auban, O., Zboray, R., Paladino, D., "Investigation of large-scale gas mixing and stratification phenomena related to LWR containment studies in the PANDA facility", *Nuclear Eng. Design,* 237(4), 409-419 (2007).

Ref. 3: Bentaïb, A. Bleyer, A., Charlet, A., Malet, F., Djebaïli-Chaumeix, N., Cheikhvarat, H., Paillard, C.E. (2005), “Experimental and numerical study of flame propagation with hydrogen gradient in a vertical facility: ENACCEF,” European Review Meeting on Severe Accident Research, Aix-en-Provence, November 14-16, 2005.

Ref. 4: Bentaib, A., Bleyer, A., Malet, J., Caroli, C., Vendel, J.; Kudriakov, S., Dabbene, F., Studer, E., Beccantini, A., Magnaud, J.P., Paillere, H. (2006), “Containment thermal-hydraulic simulations with an LP-CFD approach: Qualification matrix of the tonus code,” Fourteenth International Conference on Nuclear Engineering, Proceedings, ICONE 14.

Ref. 5: Breitung, W., Dorofeev, S., Kotchourko, A., Redlinger, R., Scholtyssek, W., Bentaib, A., L'Heriteau, J.-P., Pailhories, P., Eyink, J., Movahed, M., Petzold, K.-G., Heitsch, M., Alekseev, V., Denkevits, A., Kuznetsov, M., Efimenko, A., Okun, M.V., Huld, T., Baraldi, D. (2005), “Integral large scale experiments on hydrogen combustion for severe accident code validation-HYCOM,” Nuclear Engineering and Design, v 235, February, 2005, p 253-270.

Ref. 6: Brun, P. Cornet, J. Malet, B. Menet, E. Porcheron, J. Vendel, M. Caron-Charles, J.J. Quillico, H. Paillère and E. Studer (2002), “Specification of International Standard Problem on Containment Thermal-Hydraulics ISP-47, Step 1: TOSQAN–MISTRA,” Institut de Radioprotection et de Sureté Nucléaire and Comissariat à l’Energie Atomique (IRSN), Saclay, France.

Ref. 7: Caron-Charles, M., Quillico, J.J. Brinster, J. (2002). “Steam condensation experiments by the MISTRA facility for field containment code validation,” International Conference on Nuclear Engineering, Proceedings, ICONE, v 3, p 1041-1055

Ref. 8: Kljenak, Ivo (Jozef Stefan Institute); Babic, Miroslav; Mavko, Borut; Bajsic, Ivan (2006) “Modeling of containment atmosphere mixing and stratification experiment using a CFD approach,” Nuclear Engineering and Design, v 236, n 14-16, August, 2006, p 1682-1692.

Ref. 9: Mueller-Dietsche, W., Katzenmeier, G. (1985), “Reactor Safety Research At The Large Scale Facility HDR,” Nuclear Engineering and Design, v 88, n 3, Oct, 1985, p 241-251.

Ref. 10: Scheuerer, M., Heitsch, M., Menter, F., Egorov, Y., Toth, I., Bestion, D., Pigny, S., Pail-lere, H., Martin, A., Boucker, M., et al., (2005), “Evaluation of Computational Fluid Dynamic Methods for Reactor Safety Analysis (ECORA)”, Nuclear Engineering and Design 235, 359-368.

Ref. 11: Scholl, K.H. (1983), “Research At Full-Scale: The HDR Programme,” Nuclear Engineering International, v 28, n 336, Jan, 1983, p 39-43

Ref. 12: Studer, E., Galon, P. (1997), “Hydrogen combustion loads - Plexus calculations,” Nuclear Engineering and Design, v 174, n 2, Oct 4, 1997, p 119-134.

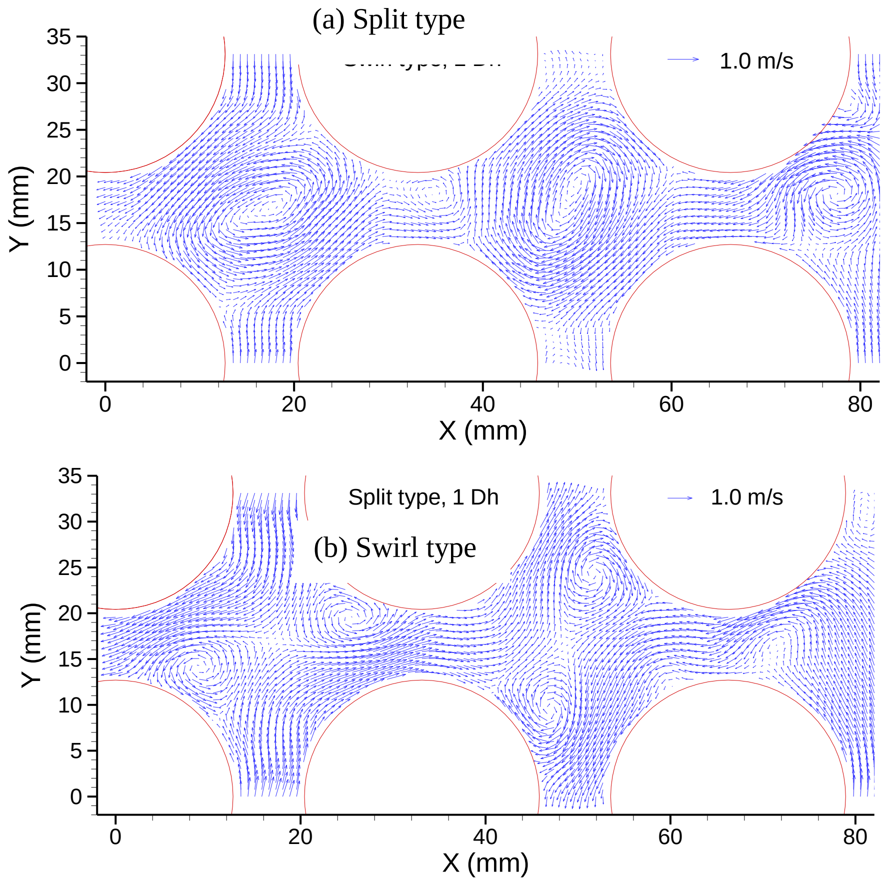
Ref. 13: Yadigaroglu, G., Dreier, J., (1998), “Passive advanced light water reactor design and the ALPHA programme at the Paul Scherrer Institute”. Kerntechnik 63, 39.

7.3.3 Possible Core-Flow Benchmarks

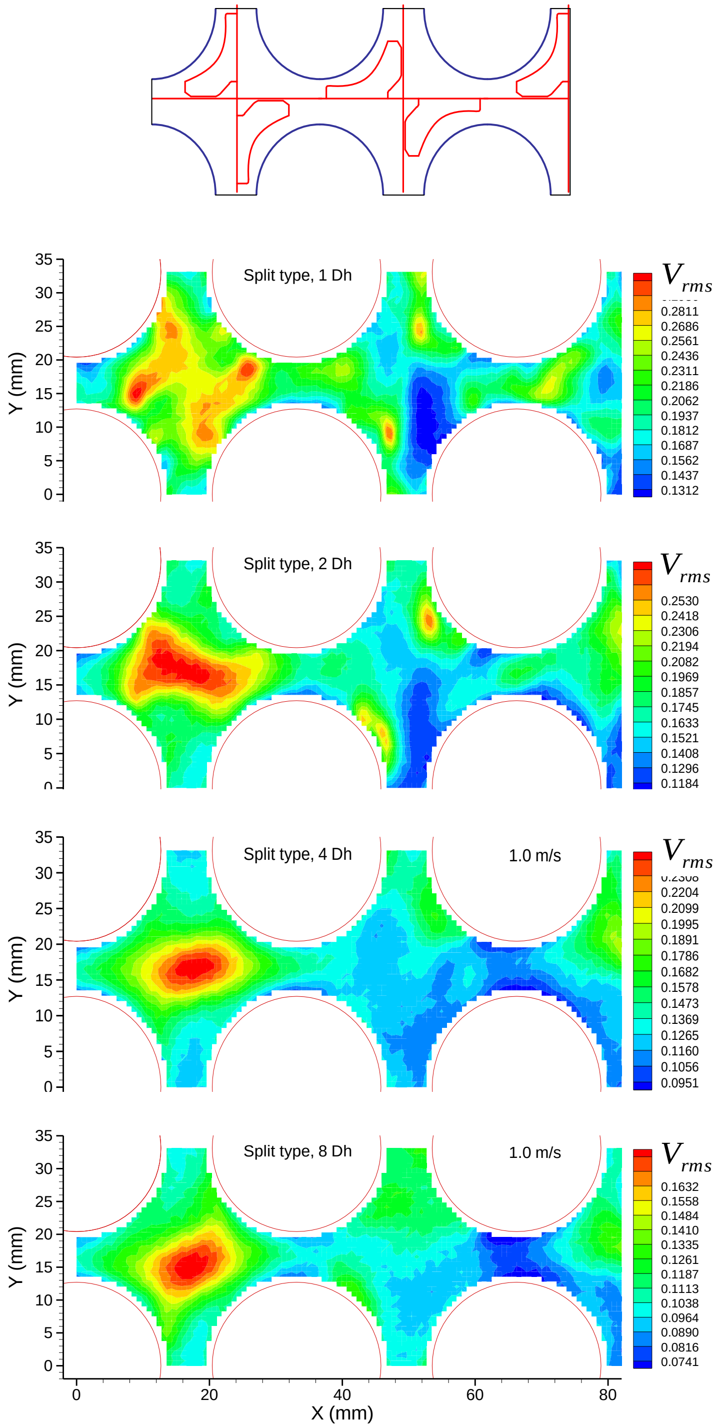
MATIS-H

This is an experimental study of detailed turbulent flow structures in horizontal square sub-channel geometry with typical mixing devices. For the fine-scale examination of the lateral flow structure on sub-channel geometry, the size of the 5x5 rod bundle array was enlarged 2.6 times from that of the real bundle. A 2-D LDA device was installed in front of the main flow cross-section of the 5x5 rod bundle array for measuring the lateral velocity components on all the sub-channels. The axial velocity component was also measured by changing the position of the LDA probe. Two spacer grids were installed to the rod bundle array. The first spacer grid, which is placed upstream of the test section, has no mixing devices, and is for the stabilization of the flow. The second spacer grid is placed at a distance 70 D_h from the first spacer grid in the downstream direction. This second spacer grid has mixing devices and causes lateral mixing and/or swirling flow. The mixing devices used in this study were typical split-type and swirl-type, respectively. A set of spacer grids can be moved in the axial direction, according to the test conditions. The experiments were performed at conditions corresponding to Re=50,000 (axial bulk velocity 1.5m/s) in the test section, and the water loop was maintained at the conditions of 35ºC and 1.5 bar during operation.

As results of detailed examinations, distinct intrinsic flow features were observed according to the type of mixing devices. For the typical split-type mixer, there was no noticeable swirling within the sub-channels, and the lateral flow was dominant in the gaps. For the swirl-type mixer, one single vortex was dominant within the sub-channel and there was relatively little lateral flow in the gaps. Lateral turbulent flow characteristics caused by the mixing devices were discussed by comparing against the bare rod experimental data. It is expected that the detailed measurement data within the sub-channels in this study can be used for the verification of related CFD codes. For this purpose, it is intended to repeat the KAERI experiments with generic rather than prototype spacer designs (to avoid problems in regard to the release of proprietary information) under the MATIS-V program with a vertical test section under both single phase and two-phase flow conditions.



Lateral velocity vectors at 1 D_h from the spacer grid



Decay of turbulence intensity along the downstream (V-component, Split type)

Ref. 1: Seok Kyu Chang, Yeun Jun Choo, Sang Ki Moon and Chul Hwa Song, "COMPARISON OF PIV AND LDV CROSSFLOW MEASUREMENTS IN SUBCHANNELS WITH VANED SPACE GRID", 12th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-12), Sheraton Station Square, Pittsburgh, Pennsylvania, U.S.A. September 30-October 4, 2007.

Ref. 2: Yang, S. K. and Chung, M. K. (1998). "Turbulent Flow through Spacer Grids in Rod Bundles," *J. Fluid Engineering, Transactions of the ASME*, Vol. 120, pp. 786-791.

Ref. 3: Shen, Y. F. *et al.* (1991). "An Investigation of Cross-flow Mixing Effect Caused by Grid Spacer with Mixing Blades in a Rod Bundle," *Nuclear Engineering Design*, 125, pp. 111-119.

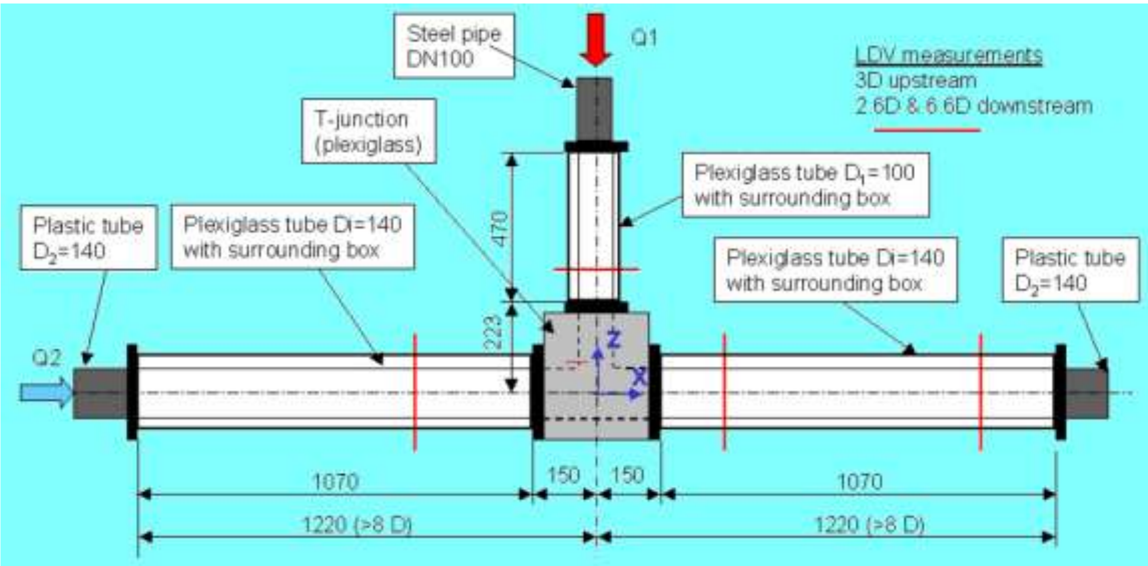
Ref. 4: Karoutas, Z., Gu, C. Y. and Scholin, B. (1995). "3-D Flow Analysis for Design of Nuclear Fuel Spacer," *Proceedings of the 7th Int. Meeting on Nuclear Reactor Thermal-Hydraulics*, New York, Sept. 10-15, Vol. 4, pp. 3153-3174.

Ref. 5: McClusky, H. L. *et al*. (2002). "Development of Swirling Flow in a Rod Bundle Sub-channel," *J. of Fluids Engineering*, Vol. 124, pp. 747-755.

7.4 OECD/NEA-Sponsored CFD Benchmarking Exercises

<u>*OECD-VATTENFALL BENCHMARK*</u>

At a meeting of the chairmen of the NEA CFD Writing Groups in 2008, it was decided to utilize the organization within the Special CFD Group of WGAMA to launch the first of a series of international benchmark exercises. Both single-phase and two-phase flow options were considered. It was generally agreed that it would be desirable to have the opportunity of setting up a blind benchmarking activity, in which participants would not have access to measured data, apart from what was necessary to define initial and boundary conditions for the numerical simulation. This would entail finding a completed, or nearly 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, and for the subsequent synthesis of the results.



Experiments to study mixing in T-junctions had been conducted at a number of facilities in France, Germany, Sweden, Japan and Switzerland, but previously unreleased test data became available from tests carried out at the Älvkarleby Laboratory of Vattenfall Research and Development in Sweden in November 2008. These became the basis of the first blind CFD benchmarking exercise to be organised within the OECD-sponsored CFD activity.

Interest in mixing in T-junctions increased following the incident at the Civaux-1 plant in France in 1998 in which both circumferential and longitudinal cracks appeared near a T-junction in the Residual

Heat Removal (RHR) system of the N4-type PWR. The Vattenfall experiment (Fig. above) was an ideal test basis for launching a blind CFD benchmarking exercise based on this safety issue. The reasoning is as follows:

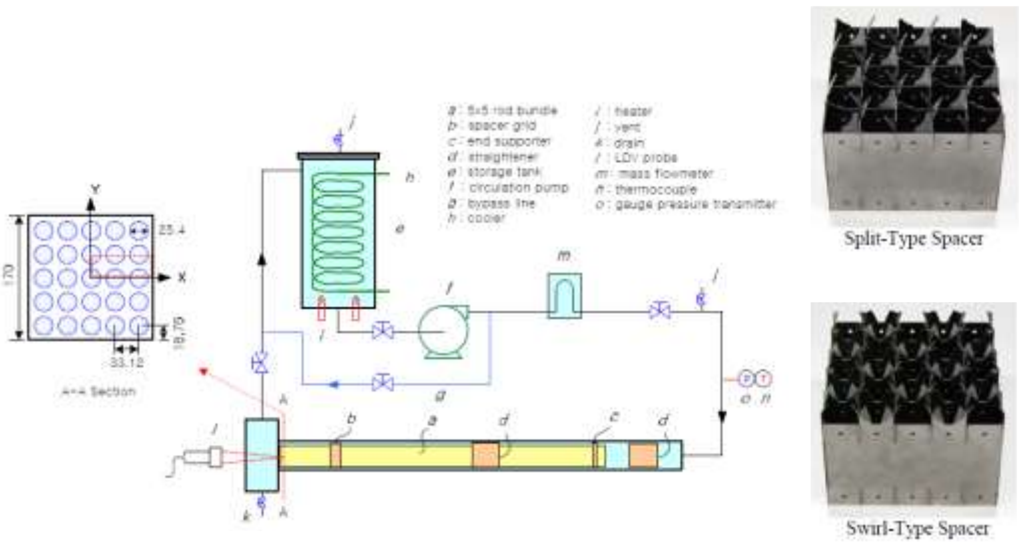
- widespread interest in high-cycle thermal fatigue had already been identified by WG2 [50];
- downstream data from the test had previously not been released;
- temperatures, velocities and turbulence data upstream had been carefully measured to provide
- precise boundary conditions for a CFD simulation [54,55];
- uncertainty estimates were available for all measurements.

Vattenfall R&D agreed to release measured data to all those who submitted blind calculations to this benchmarking exercise.

The activity ran from May 2009 (Kick-Off Meeting) to December 2010. In total, 29 participants submitted blind numerical predictions for synthesis. A full CSNI report is available on the NEA website.

OECD-KAERI BENCHMARK

This activity focuses on the ability of CFD codes to predict turbulence characteristics downstream of a spacer grid in a core channel geometry. The experiment is based on a special test performed under isothermal conditions in a horizontal rod bundle configuration in the MATiS-H cold-flow facility at the Korea Atomic Energy Research Institute (KAERI), carried out in early Spring, 2012.



Two spacer grids (of generic design), of the split type and swirl-type, were involved in the study. Computer Aided Design (CAD) files of the spacer grids were made available by KAERI to aid CFD mesh generation. The benchmark was launched in April 2011, and blind predictions collected one year later. A synthesis report has been written, and was presented at the CFD4NRS-4 Workshop in September 2012. In addition, a full CSNI report on the entire activity has just been approved and will be distributed in 2013.

8. CONCLUSIONS AND RECOMMENDATIONS

The use of computational methods for performing safety analyses of reactor systems has been established for nearly 40 years. Very reliable codes have been developed for analysing the primary system in particular, and results from these analyses are often used in the safety assessment of nuclear power systems undertaken by the regulatory authorities. Similarly, but to a lesser extent, programs have also been written for containment and severe accident analyses. 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 treat flows of this type, and to handle geometries of almost arbitrary complexity. Already, CFD has been successfully applied to such flows, and to a limited extent has made up for a lack of applicable test data in better quantifying safety margins. Consequently, CFD 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 range of validity is well known, and controlled internally within the numerical algorithms. Herein lies the trustworthiness of the numerical predictions of such 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 NRS applications has therefore to be addressed before the use of CFD can be considered as trustworthy. This issue represented the primary focus of the work carried out by the second of the OECD/NEA Writing Groups (WG2), its findings, appropriately updated as a consequence of further information produced by members of the CFD Task Group created by WGAMA, are embodied in the present document.

The document provides a list of NRS problems for which it is considered CFD analysis is required, or its application is expected to result in positive benefits in terms of better understanding and improved safety margins. The list contains safety issues of relevance to fluid flows in the core, primary circuit and containment, both under normal and abnormal operating conditions, and during accident sequences. The list contains both single-phase and two-phase safety items, 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, NEA/CSNI/R(2007)15 (update in preparation).

Recognising that CFD is already an established technology outside of the nuclear community, a list of the existing assessment bases from other application areas has also been included, and their relevance to NRS issues discussed. It is shown that these databases are principally of two types: those concerned with aspects of trustworthiness of CFD code predictions in general industrial applications (ERCOFTAC, QNET-CFD, FLOWNET), and those focussed on specialised topics (MARNET, NPARC, AIAA). The usefulness and relevance of these databases to NRS has been assessed. In addition, most CFD codes currently being used for NRS analysis have their own, custom-built assessment bases, the data being provided from both within and external to the nuclear community. It was concluded that application of CFD to NRS problems can benefit indirectly from these databases, and the continuing efforts to extend them, but that a well-maintained, NRS-specific database would be a valuable addition.

Descriptions of the existing CFD assessment bases that have been established specifically within the nuclear domain have been listed here, and their usefulness discussed. Typical examples are experiments devoted to the boron dilution and in-vessel mixing issues (ISP-43, ROCOM, Vattenfall 1/5th Scale Benchmark, UPTF TRAM C3,), pressurised thermal shock (UPTF TRAM C2), and thermal fatigue in pipes (THERFAT, Forsmarks), all of which have already been the subject of benchmarking activities. Details of where this information may be obtained has been given, in particular the EU Framework Programmes, such as ASTAR, ECORA, EUBORA, FLOWMIX-R and ASCHLIM, which have provided direct NRS-specific data and/or have each focused on relevant aspects of the CFD modelling.

The technology gaps which need to be closed to make CFD a more trustworthy analytical tool have also been identified. These include, for example, lack of a proper uncertainty methodology; limitations in the range of application of turbulence models, for example in stratified and buoyant flows; coupling of CFD with neutronics and system codes, needed to keep simulations to a manageable size; and generally computer power limitations in simulating long transients. In each case, a discussion is given of the relevance and importance of the problem to NRS analysis, what has been achieved to date, and what still needs to be done in the future. Particular application areas for which CFD simulations need to be improved are in stratified flows, containment modelling, aerosol transport and deposition and liquid-metal heat transfer. In other areas, such as in-vessel mixing, the models may be adequate but grid resolution is inadequate due to the current lack of machine power, a situation that will certainly improve with time.

This last point, the computational overhead of performing CFD simulations in comparison with system code transient computations, may still be regarded as a definite limitation of the potential for directly using CFD in licensing procedures, even for single-phase applications for which the underlying models are well-established. The uncertainty quantification methodology for system codes generally requires 50-100 computations to be carried out, and the statistical method of Latin Hypercube Sampling (LHS) is becoming widespread in order to optimise the efficiency of the random parameter sampling. This cannot, at the present time, be mirrored with CFD, and until it can a different methodology needs to be established. However, in the spirit of BPGs, at least mesh-independency must be demonstrated, and some limited study of sensitivity to input parameters should be attempted. The issue of access to the source code of CFD software, particularly in regard to the commercial codes, will also have to be addressed before CFD is accepted as an analysis tool by the regulatory bodies.

There is a distinct lack of quality validation data for aerosol transport, even though the phenomenon was identified as a key process in containment modelling, and one that can only be treated mechanistically by the use of CFD. The experiments carried out at the PHEBUS facility as part of the EU 5th Framework Programme PHEBEN produced only data of an integral nature, and as such very limited in regard to validating CFD models. Comprehensive, local aerosol deposition data appear only to be available for pipes (straight and elbowed), and for some non-nuclear applications, such as atmospheric pollution. This is one key area where future CFD assessment needs to be focused.

Important new information has been provided by the material presented at the CFD4NRS series of Workshops, in which numerical simulations with a strong emphasis on validation were particularly encouraged, and the reporting of experiments which provided high-quality data suitable for CFD validation. Participation in the workshops has enabled a list of existing databases to be assembled of possible candidates for future benchmarking activities for: (1) primary circuits, (2) core-flow regions, and (3) containments, for which data of the type needed for CFD benchmarking already exists, or is likely to be available in the near future.

This updated document represents a continuing process in establishing an assessment database for the application of CFD to NRS problems, but in many places reflects the time and manpower restrictions imposed on the authors by their parent organisations, and considerable further work still needs to be done in terms of both presentation and technical content. Sections of the report remain unbalanced in terms of detail, reflecting not only the subjective inputs of the authors, but whether the safety issue being addressed is of a country-specific nature or of more common concern; the level of detail is higher in the latter case, and with better perspectives. Part of the recommended obligation to regularly update this document must include an attempt to equilibrate the information level. In addition, similar information appears in different sections of the report. This was done to avoid excessive page-turning or scrolling, but gives the document an appearance of disjointedness if read in a continuous manner. The updates to the original WG2 report contained in the present document have not rectified these defects. A more efficient method of control would be to install hyperlinks to the web-based version of the document, as recommended below.

CFD remains a very dynamic technology, and with its increasing use within the nuclear domain there will be ever greater demands to document current capabilities and prove their trustworthiness by means of validation exercises. It is therefore expected that any existing list of specific assessment databases will soon require further updating. To prevent the important information assembled in this document from becoming obsolete, the following recommendations are made.

- Extend the process of consolidating the information contained here through continuous updating of the web-based version of the WG2 document. This process is necessary to ensure that the NRS assessment database is readily accessible to all, topical, and as dynamic and mobile as the CFD technology itself.

- The forum for numerical analysts and experimentalists to exchange information in the field of NRS-related activities relevant to CFD validation provided by the series of CFD4NRS workshops should continue, thus providing a continuous source of information to build into the web-based assessment matrix.

- New blind CFD benchmarking exercises should be defined, both to encourage the release of previously restricted CFD-grade data from experiments, to test the skills of the CFD practitioners, and perhaps persuade the software developers to improve their models, where these have proved lacking. To this end, it is encouraging to note that representatives of the large commercial software houses actively participate in the benchmarks.

- The Special CFD Group, which was first set up within WGAMA in 2007, and initially comprised the chairmen of the original three Writing Groups (together with the NEA secretariat), can continue to act as the central organising body for the above activities, provided new members are appointed to replace the “old guard”. The time-scale for this process is (i) *overdue* for the WG1 chairman (J. H. Mahaffy reached pensionable age in 2009); (ii) *imminent* for the WG2 chairman (B. L. Smith reached pensionable age in 2012); and *within sight* for the WG3 chairman (D. Bestion will reach pensionable age in 2017). It is important to ensure a smooth transition to a new group membership before the existing expertise is lost.

APPENDIX 1: OECD-IAEA WORKSHOPS IN THE CFD4NRS SERIES

CFD4NRS: Benchmarking of CFD Codes for Application to Nuclear Reactor Safety

Background

Computational Fluid Dynamics (CFD) is to an increasing extent being adopted in nuclear reactor safety analyses as a tool that enables specific safety relevant phenomena occurring in the reactor coolant system to be better described. The Committee on the Safety of Nuclear Installations (CSNI), which is responsible for the activities of the Nuclear Energy Agency that support advancing the technical base of the safety of nuclear installations, has in recent years conducted an important activity in the CFD area. This activity has been carried out within the scope of the CSNI working group on the analysis and management of accidents (GAMA), and has mainly focused on the formulation of user guidelines and on the assessment and verification of CFD codes. It is in this WGAMA framework that the present workshop was organized and carried out.

Computational methods have supplemented scaled model experiments, and even prototypic tests, in the safety analysis of reactor systems for nearly 30 years. During this time, very reliable codes have been developed for analysing the primary system, and similar programs have also been written for containment and severe accident analyses. However, many traditional reactor system and containment codes are modelled as networks of 1-D or even 0-D cells. It is evident, however, that the flow in components such as the upper and lower plena, downcomer and core of a reactor vessel is essentially 3-D in nature. Natural circulation, mixing and stratification in containments is also 3-D, and representing such complex flows by pseudo 1-D or 0-D approximations may lead to erroneous, and not necessarily conservative, conclusions. CFD has the potential to handle geometries of arbitrary complexity, and is poised to fill this technology gap for single-phase applications, though considerable further development of closure relations will be necessary before multi-phase Nuclear Reactor Safety (NRS) applications may be approached with confidence using CFD.

Traditional approaches to NRS analysis using system codes for example have been successful because a very large database of phasic exchange correlations has been built into them. The correlations have been formulated from essentially 1-D special-effects experiments, and their range of validity well scrutinised. Data on the exchange of mass, momentum and energy between phases for 3-D flows is very sparse in comparison. Thus, although 1-D formulations may restrict the use of system codes in simulations in which there is complex geometry, the physical models are well-established and reliable, provided they are used within their specified ranges of validity. The trend has therefore been to continue with such approaches, and live within their geometrical limitations.

For containment issues, lumped-parameter codes include models for system components, such as recombiners, sprays, sumps, etc., which enable realistic simulations of accident scenarios to be undertaken without excessive computational costs. To take into account such systems in a multi-dimensional (CFD) simulation remains a challenging task, and attempts to do this have only recently begun, and these in dedicated CFD codes rather than in commercial, general-purpose CFD software.

The issue of the validity range of CFD codes for 3-D NRS applications has to be addressed before the use of CFD may be considered as routine and trustworthy as it is for example in the turbo-machinery, automobile and aerospace industries. However, the application of CFD methods to NRS-related issues is not straightforward. In many cases, even for single-phase problems, nuclear thermal-hydraulic flows lie outside the range of current computer capacity, especially in the case of long, evolving transient flows with strong heat transfer.

These issues were discussed in the group of experts designated by CSNI/WGAMA to carry out the task of establishing an assessment matrix for CFD application to NRS, concentrating on single-phase phenomena. As part of this process, it was decided to organise an international workshop to promote the availability and distribution of experimental data suitable for NRS benchmarking, and to monitor the current status of CFD validation exercises relevant to NRS issues. The workshop would also cover two-phase aspects, and if the venture was successful, organisation of further workshops on this theme was envisaged.

Scope and Objectives

The purpose of the workshop was to provide a forum for numerical analysts and experimentalists to exchange information in the field of NRS-related activities relevant to CFD validation, with the objective of providing input to WGAMA CFD experts to create a practical, state-of-the-art, web-based assessment matrix on the use of CFD for NRS applications.

Numerical simulations with a strong emphasis on validation were welcomed in such areas as heat transfer, buoyancy, stratification, natural circulation, free-surface modelling, turbulent mixing and multi-phase flow. These would relate to such NRS-relevant issues as: pressurized thermal shocks, boron dilution, hydrogen distribution, induced breaks, thermal striping, etc. The use of systematic error quantification and Best Practice Guidelines was encouraged.

Papers reporting experiments providing high-quality data suitable for CFD validation, specifically in the area of NRS, were given high priority. Here, emphasis was placed on the availability of local measurements, especially multi-dimensional velocity measurements obtained using such techniques as laser-doppler velocimetry, hot-film/wire anemometry, particle image velocimetry, laser induced fluorescence, etc. A particular point of scrutiny for papers in this category was whether an assessment of error bounds and measurement uncertainties was included.

Welcoming Address

L. Hahn (GRS, Germany)

Invited Lectures

1. *M. Réocreux (IRSN, France)*
   Safety Issues Concerning Nuclear Power Plants: The Role of CFD
2. *M. Gavrilas (NRC, USA)*
   Lessons Learned from International Standard Problem No. 43 on Boron Mixing
3. *W. Oberkampf (SNL, USA)*
   Design of and Comparison with Verification and Validation Benchmarks
4. *H-M.Prasser (HZDR, Germany/ETHZ, Switzerland)*
   Novel Experimental Measuring Techniques Required to Provide Data for CFD Validation
5. *G. Yadigaroglu (ASCOMP/ETHZ, Switzerland)*
   CFD4NRS with a Focus on Experimental and CMFD Investigations of Bubbly Flows

Technical Session A1

Plant Applications

1. *M. Böttcher*
   Detailed CFX-5 Study of the Coolant Mixing within the Reactor Pressure Vessel of a VVER-1000 Reactor during a Non-Symmetrical Heat-Up Test
2. *I. Boros, A. Aszódi*
   Analysis of Thermal Stratification in the Primary Circuit with the CFX Code
3. *E. Romero*
   CFD Modelling of a Negatively Buoyant Purge Flow in the Body of a Reactor Coolant Circulator
4. *G. Légrádi, I. Boros, A. Aszódi*
   Comprehensive CFD Analyses Concerning the Serious Incident which occurred in the PAKS NPP in Spring 2003

Technical Session B1

Advanced Reactors

5. *T. Morii*
   Hydraulic Flow Tests of APWR Reactor Internals for Safety Analysis
6. *R. W. Johnson*
   Modeling Strategies for Unsteady Turbulent Flows in the Lower Plenum of the VHTR
7. *H. S. Kang, C. H. Song*
   CFD Analysis of Thermal Mixing in a Subcooled Water Pool under High Steam Mass Flux
8. *K. Velusamy, K. Natesan, P. Selvaraj, P. Chellapandi, S. C. Chetal, T. Sundararajan, S. Suyambazhahan (WITHDRAWN)*
   CFD Studies in the Prediction of Thermal Striping in an LMFBR

Technical Session A2

Benchmark Exercises

9. *M. Andreani, K. Haller, M. Heitsch, B. Hemström, I. Karppinen, J. Macek, J.Schmid, H. Paillere, I. Toth*
   A Benchmark Exercise on the use of CFD Codes for Containment Issues using Best Practice Guidelines: a Computational Challenge
10. *T. Toppila*
    CFD Simulation of FORTUM PTS Experiment

Technical Session B2

CANDU Reactors

11. *H. S. Kang*
    CFD Analysis for the Experimental Investigation of a Single Channel Post-Blowdown
12. *T. Kim, B. W. Rhee, J. H. Park*
    CFX Simulation of a Horizontal Heater Rods Test

Technical Session A3

Novel Applications

13. *U.Graf, P.Papadimitriou*
    Simulation of Two-Phase Flows in Vertical Tubes with the CFD Code FLUBOX
14. *Y. A. Hassan*
    Large-Eddy Simulation in Pebble Bed Gas Cooled Core Reactors

Technical Session B3

Containment Issues I

15. *Kljenak, M. Babić, B. Mavko*
    Prediction of Light Gas Distribution in Containment Experimental Facilities using CFX4 Code: Jozef Stefan Institute Experience
16. *S. Kudriakov, F. Dabbene, E. Studer, A. Beccantini, J.P. Magnaud, H. Paillère, A. Bentaib, A. Bleyer, J. Malet, C. Caroli*
    The TONUS CFD Code for Hydrogen Risk Analysis: Physical Models, Numerical Schemes and Validation Matrix

Technical Session A4

Boron Dilution

17. *S. Kliem, T. Sühnel, U. Rohde, T. Höhne, H.-M. Prasser, F.-P. Weiss*
    Experiments at the Mixing Test Facility ROCOM for Benchmarking of CFD Codes
18. *T. V. Dury, B. Hemström, S. V. Shepel*
    CFD Simulation of the Vattenfall 1/5th-Scale PWR Model for Boron Dilution Studies
19. *E. Graffard, F. Goux*
    CFX Code Application to the French Reactor for Inherent Boron Dilution Safety Issue

Technical Session B4

Containment Issues II

20. *E. Porcheron, P. Lemaitre, A. Nuboer, V. Rochas, J. Vendel*
    Experimental Study of Heat, Mass and Momentum Transfers in a Spray in the TOSQAN Facility

21. *J. Malet, P. Lemaitre, E. Porcheron, J. Vende*[1]*, L. Blumenfeld, F. Dabbene, I. Tkatschenko*
    Benchmarking of CFD and LP Codes for Spray Systems in Containment Applications: Spray Tests at Two Different Scales in the TOSQAN and MISTRA Facilities
22. *M. Houkema, N.B. Siccama*
    Validation of the CFX-4 CFD Code for Containment Thermal-Hydraulics

Technical Session A5

Mixing in Primary Circuit

23. *T. Höhne, S. Kliem*
    Coolant Mixing Studies of Natural Circulation Flows at the ROCOM Test Facility using ANSYS CFX
24. *S. K. Chang, S. K. Moon, B. D. Kim, W. P. Baek, Y. D. Choi*
    Phenomenological Investigations on the Turbulent Flow Structures in a Rod Bundle Array with Mixing Devices
25. *J. Westin, F. Alavyoon, L. Andersson, P. Veber, M. Henriksson, C. Andersson*
    Experiments and Unsteady CFD-Calculations of Thermal Mixing in a T-junction

Technical Session B5

Containment Issues III

26. *P. Royl, J. R. Travis, W. Breitung*
    Modelling and Validation of Catalytic Hydrogen Recombination in the 3D CFD Code GASFLOW II
27. *H. Wilkening, D. Baraldi, M. Heitsch*
    On the Importance of Validation when using Commercial CFD Codes in Nuclear Reactor Safety
28. *R. Redlinger*
    DET3D - A CFD Tool for Simulating Hydrogen Combustion in Nuclear Reactor Safety

Technical Session A6

Stratification Issues

29. *T. Wintterle, E. Laurien, T. Stäbler, L. Meyer, T. Schulenberg*
    Experimental and Numerical Investigation of Counter-Current Stratified Flows in Horizontal Channels
30. *L. Štrubelj, I. Tiselj, B. Končar*
    Modelling of Direct Contact Condensation in Horizontally Stratified Flow with CFX Code
31. *C. Vallée, T. Höhne, H.-M. Prasser, T. Sühne*
    Experimental Investigation and CFD Simulation of Horizontal Stratified Two-Phase Flow Phenomena

Technical Session B6

Code Validation

32. *Th. Frank, P.J. Zwart,E. Krepper, H.-M. Prasser, D. Lucas*
    Validation of CFD Models for Mono- and Polydisperse Air-Water Two-Phase Flows in Pipes
33. *V.Ustinenko, M.Samigulin, A.Ioilev, S.Lo, A.Tentner, A.Lychagin, A.Razin, V.Girin, Ye.Vanyukov*
    Validation of CFD-BWR: a New Two-Phase Computational Fluid Dynamics Model for Boiling Water Reactor Analysis
34. *U. Bieder, E. Graffard*
    Qualification of the CFD Code TRIO_U for Full-Scale Nuclear Reactor Applications

Technical Session A7

Boiling Models

35. *B. J. Yun, D. J. Euh, C. H. Song*
    Experimental Investigation of Subcooled Boiling on One Side of a Heated Rectangular Channel
36. *S. Mimouni, M. Boucker, J. Laviéville, D. Bestion*
    Modeling and Computation of Cavitation and Boiling Bubbly Flows with the NEPTUNE_CFD Code
37. *B. Končar, E. Krepper*
    CFD Simulation of Forced Convective Boiling in Heated Channels

Technical Session B7

Containment Issues IV

38. *P. Royl, U. J. Lee, J. R. Travis, W. Breitung*
    Benchmarking of the 3D CFD Code GASFLOW II with Containment Thermal Hydraulic Tests from HDR and ThAI
39. *A. Dehbi*
    Assessment of a New FLUENT Model for Particle Dispersion in Turbulent Flows

Conclusions and Recommendations

There were 98 registered participants to the workshop to hear 5 invited talks and 39 technical papers. This is perhaps a good measure of the level of general interest in the workshop. The messages coming back to the organisers from the participants were that the workshop was well organised and that the subject material well chosen. As there was only a 60% success rate for the extended abstracts sent in to the organisers for acceptance, the quality of the papers was high, and the focus of them on the central issue strong.

The case for future workshops in the series was discussed openly during the final panel session. It was pointed out that 2/3 of the papers accepted for CFD4NRS were concerned with single-phase calculations and experiments, while 1/3 were dedicated to multi-phase issues. The ratio probably reflects the degree of maturity of CFD in the respective areas, but nonetheless suggests a growing acknowledgement of the role of multi-phase CFD in nuclear NRS issues.

Following on from this observation, CEA proposed a follow-up meeting, perhaps hosted by CEA Grenoble, in which the ratio of single-phase to two-phase papers would be inverted, and would expand the area of advanced instrumentation needed for providing local data needed to validate the models currently being proposed for multi-phase CFD. The suggestion received encouraging remarks from the audience. It was also generally agreed that the frequency of future workshops should be 2-3 years, allowing sufficient time for the technology to advance, and minimise the chance of overlap with the material presented at CFD4NRS.

The Organising and Scientific Committees had discussed at an early stage whether the editor of an appropriate archival journal should be approached in regard to offering publication of selected papers from the workshop in a special issue of the journal. On balance, it was considered that it would be too great a risk to an editor for a first-of-a-kind conference with an untried format. It therefore came as a bonus that Professor Yassin Hassan, co-editor of Nuclear Engineering and Design, and a participant at CFD4NRS, would make just this suggestion. The offer has been followed up, and some 25 authors of technical papers and 3 invited speakers have expressed interest in this proposal. Again, the offer reflects the high quality of the presented material, and the general level of interest in what the workshop aimed to achieve. It is anticipated that the special issue of NED dedicated to CFD4NRS will appear in 2008.

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

- Best Practice Guidelines should be followed as far as practical 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 used for code validation should include estimates of measurement uncertainties, and should include detailed information concerning initial and boundary conditions.
- Experimenters involved in producing data for validating CFD models and/or applications should collaborate actively with CFD practitioners in advance of setting up their instrumentation. This interface 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 measurements against which to compare code predictions) is optimal, and the frequency of data acquisition is appropriate to the time-scale(s) of significant fluid-dynamic/heat-transfer/phase-exchange events.
- This workshop proved to be a very valuable means to assess the status of CFD code validation and application. Specialised workshops of this type should be organised at suitable time intervals also in the future, in order to maintain continuity, monitor progress, and exchange experiences on CFD code validation and applications.

XCFD4NRS: Experiments and CFD Applications to Nuclear Reactor Safety

Background

Computational Fluid Dynamics (CFD) is to an increasing extent being adopted in nuclear reactor safety analyses as a tool that enables specific safety relevant phenomena occurring in the reactor coolant system to be better described. The Committee on the Safety of Nuclear Installations (CSNI), which is responsible for the activities of the OECD Nuclear Energy Agency that support advancing the technical base of the safety of nuclear installations, has in recent years conducted an important activity in the CFD area. This activity has been carried out within the scope of the CSNI working group on the analysis and management of accidents (WGAMA), and has mainly focused on the formulation of user guidelines and on the assessment and verification of CFD codes. It is in this WGAMA framework that a first workshop, CFD4NRS, was organized and held in Garching, Germany in 2006.

Following the success of the first workshop, XCFD4NRS was intended to extend the forum created for numerical analysts and experimentalists to exchange information in the field of Nuclear Reactor Safety (NRS) related activities relevant to Computational Fluid Dynamics (CFD) validation, but this time with more emphasis placed on new experimental techniques and two-phase CFD applications.

Scope and Objectives

The purpose of the workshop was to provide a forum for numerical analysts and experimentalists to exchange information in the field of NRS-related activities relevant to CFD validation, with the objective of providing input to WGAMA CFD experts to create a practical, state-of-the-art, web-based assessment matrix on the use of CFD for NRS applications.

The scope of XCFD4NRS includes:

- Single-phase and two-phase CFD simulations with an emphasis on validation in areas such as: boiling flows, free-surface flows, direct contact condensation and turbulent mixing. These applications should relate to NRS-relevant issues such as: pressurized thermal shocks, critical heat flux, pool heat exchangers, boron dilution, hydrogen distribution, thermal striping, etc. Discussion of validation of the CFD tool, use of systematic error quantification and Best Practice Guidelines (BPGs) was encouraged and considered in the paper review process.

- Experiments providing data suitable for CFD validation, specifically in the area of NRS. These should focus on local measurements using multi-sensor optical or electrical probes, laser-doppler velocimetry, hot-film/wire anemometry, particle image velocimetry and laser induced fluorescence. Papers should include a discussion of measurement uncertainties.

Welcoming Address

C. Chauliac (CEA, France)

Invited Lectures

1. *V. Teschendorf (GRS, Germany)*
   The Role of CFD in NPP Safety
2. *Y. Hassan (Texas A&M, USA)*
   Single Phase CFD Simulation and Experimental Validation for Advanced Nuclear System Components
3. *T. Hibiki (Purdue Univ., USA)*
   Modelling and Measurement of Interfacial Area Concentration in Two-phase Flow
4. *S. Banerjee (City University of New York, USA)*
   Advanced Fine-Scale Modelling of Two-Phase Flow
5. *T. Schulenberg (KIT, Germany)*
   Experimental Techniques for Heavy Liquid Metals

Summaries of the Activities of WGAMA Writing Groups on CFD

6. *J. H. Mahaffy (PSU, USA)*
   Best Practice Guidelines for the use of CFD for NRS Applications
7. *B. L. Smith(PSI, Switzerland)*
   Assessment of CFD for NRS
8. *D. Bestion (CEA, France)*
   Extension of CFD use to two-phase NRS issues

Technical Session HOR

Horizontal Flow - Pipe Flow

HOR-01 *Y. Bartosiewicz, J.-M. Seynhaeve, C. Vallée, T. Höhne, J.M. Laviéville*
Modelling free surface flows relevant to a PTS scenario: comparison between experimental data and three RANS based CFD-codes. Comments on the CFD-experiment integration and best practice guidelines

HOR-02 *H. Lemonnier*
Nuclear Magnetic Resonance: A new tool for the validation of multi-phase multi-dimensional CFD codes

HOR-03 *M. Marchand, M. Bottin, J.P. Berlandis, E. Hervieu*
Experimental investigation of stratification phenomena in horizontal two-phase flows for CFD validation

HOR-04 *L. Štrubelj, I. Tiselj*
Numerical modelling of direct contact condensation in transition from stratified to slug flow

HOR-05 (Poster) *C. Vallée, D. Lucas, M. Beyer, H. Pietruske, P. Schütz, H. Car*
Experimental CFD grade data for stratified two-phase flows

Technical Session AC

Accident Analysis

AC-01 *E. Krepper, G. Cartland-Glover, A. Grahn, F.P. Weiss*
Experiments and CFD-modelling of insulation debris transport phenomena in water flow
AC-02 *T. Brandt, V. Lestinen, T. Toppila, J. Kähkönen, A. Timperi, T. Pättikangas, I. Karppinen*
Fluid-structure interaction analysis of Large-Break Loss of Coolant Accident
AC-03 *C. López del Prá, F. J. S. Velasco, L. E. Herranz*
Simulation of a gas jet entering a failed steam generator during a SGTR sequence: validation of a FLUENT 6.2 model
AC-04 *C. T. Tran, P. Kudinov and T. N. Dinh*
An approach to numerical simulation and analysis of molten corium coolability in a BWR lower head
AC-05 (Poster) *Jeong Ik Lee, Soon Joon Hong, Jonguk Kim, Byung Chul Lee, Young Seok Bang, Deog Yeon Oh, Byung Gil Huh*
Experimental CFD grade data for stratified two-phase flows
AC-06 (Poster) *B.A. Gabaraev, E.K. Karasyov, O.Yu. Novoselsky, S.Z. Lutovinov, L.K. Tikhonenko, Ye.I. Trubkin, A.V. Shishov*
Data obtained at high coolant parameters suitable for validation of 3D models

Technical Session PTS

Pressurized Thermal Shock

PTS-01 *P. Coste, J. Pouvreau, J. Laviéville, M. Boucker*
Status of a two-phase CFD approach to the PTS issue
PTS-02 *T. Farkas, I. Tóth*
FLUENT analysis of a ROSA cold leg stratification
PTS-03 *H. S. Kang, Y.-J. Youn, C.-H. Song*
CFD analysis of a turbulent jet behaviour induced by a steam jet discharge through a single hole in a subcooled water pool
PTS-04 *Y. J. Choo, C.-H. Song, Y. J. Youn*
PIV measurement of turbulent jet and pool mixing produced by a steam jet in a sub-cooled water pool
PTS-05 (Poster) *M. Schmidtke, D. Lucas*
On the modelling of bubble entrainment by impinging jets in CFD simulations
PTS-06 (Poster) *V. Tanskanen, D. Lakehal, M. Puustinen*
Validation of Direct Contact Condensation CFD models against condensation pool experiment

Technical Session CO

Containment Thermal Hydraulics

CO-01 *S. Mimouni, J-S. Lamy, J. Lavieville, S. Guieu, M. Martin*
Modelling of sprays in containment applications with A CMFD code
CO-02 *P. Royl, J.R. Travis, W. Breitung, Jongtae Kim, Sang Baik Kim*
GASFLOW validation with Panda tests from the OECD SETH Benchmark covering steam/air and steam/helium/air mixtures
CO-03 *M. Ritterath, H.-M. Prasser, D. Paladino, N. Mitric*
New PANDA instrumentation for assessing gas concentration distributions in Containment Compartments

CO-04 *M. Andreani, D. Paladino, T. George*
On the unexpectedly large effect of re-vaporization of the condensate liquid film in two tests in the PANDA facility revealed by simulations with the GOTHIC code

CO-05 *M. Heitsch, D. Baraldi, H. Wilkening*
Validation of CFD for Containment Jet Flows including Condensation

CO-06 (Poster) *S. Kelm, W. Jahn, E.A Reinecke*
Operational behaviour of catalytic recombiners - experimental results and modelling approaches

CO-07 (Poster) *Jinbiao Xiong, Yanhua Yang, Xu Cheng*
Effects of spray modes on Hydrogen risk in a Chinese NPP

Technical Session MIX

Mixing Issues

MIX-01 *M. Böttcher*
Primary Loop Study of a VVER-1000 reactor with special focus on coolant mixing

MIX-02 *M. J. Da Silva, S. Thiele, T. Höhne, R. Vaibar, U. Hampel*
Experimental studies and CFD calculations for buoyancy driven mixing phenomena

MIX-03 *S. Kliem, T. Höhne, U. Rohde, F.-P. Weiss*
Experiments on slug mixing under natural circulation conditions at the ROCOM test facility using high resolution measurement technique and numerical modelling

MIX-04 *F. Ducros, U. Bieder, O. Cioni, T. Fortin, B. Fournier, P. Quéméré*
Verification and validation considerations regarding the qualification of numerical schemes for LES dilution problems

MIX-05 *S. Tóth, A. Aszódi*
CFD Study on coolant mixing in VVER-440 Fuel rod bundle and fuel assembly head

MIX-06 *H.-M. Prasser, A. Manera, B. Niceno, M. Simiano, B. Smith, C. Walker, R. Zboray*
Fluid mixing at a T-junction

MIX-07 *Th. Frank, M. Adlakha, C. Lifante, H.-M. Prasser, F. Menter*
Simulation of turbulent and thermal mixing in T-junctions using URANS and scale-resolving turbulence models in ANSYS-CFX

MIX-08 *A.K. Kuczaj, E.M.J. Komen*
Large Eddy simulation of turbulent mixing in a T-junction

MIX-09 (Poster) *M. Bykov, A. Moskalev, A. Shishov, O. Kudryavtsev, D. Posysaev*
Validation of CFD code ANSYS CFX against experiments with saline slug mixing performed at the Gidropress 4-loop WWER-1000 test facility

MIX-10 (Poster) *M. Bykov, A. Moskalev, D. Posysaev, O. Kudryavtsev, A. Shishov*
Validation of CFD code ANSYS CFX against experiments with asymmetric saline injection performed at the Gidropress 4-loop WWER-1000 test facility

Technical Session BOI

Boiling Flow, Bubbly Flow and Critical Heat Flux

BOI-01 *D. Lucas, M. Beyer, J. Kussin, P. Schütz*
Benchmark database on the evolution of two-phase flows in a vertical pipe

BOI-02 *B.J. Yun, B.U.Bae, W.M.Park, D.J.Euh, G.C.Park, C-.H. Song*
Characteristics of local bubble parameters of sub-cooled boiling flow in an annulus

BOI-03 *B. Končar, B. Mavko*
Wall-to-fluid heat transfer mechanisms in boiling flows

BOI-04 *B.U. Bae, B.J. Yun, H.Y. Yoon, G.C. Park, C.-H. Song*
Development of two-phase flow CFD code (EAGLE) with interfacial area transport equation

	for analysis of subcooled boiling flow
BOI-05	*S. Mimouni, F. Archambeau, M. Boucker, J. Lavieville, C. Morel* A second order turbulence model based on a Reynolds Stress approach for two-phase boiling flow and application to fuel assembly analysis
BOI-06	*A. Bieberle, D. Hoppe, C. Zippe, E. Schleicher, M. Tschofen, T. Suehnel, W. Zimmermann, U. Hampel* Void measurement in boiling water reactor rod bundles using high resolution gamma ray Tomography
BOI-07	*M. Damsohn, H.-M. Prasser* CFD validation of film flows by novel high speed liquid film sensor with high spatial resolution
BOI-08	*F. Fischer, U. Hampel* Ultra fast electron beam X-ray computed tomography for two-phase flow measurement
BOI-09	*M. C. Galassi, F. Moretti, F. D'Auria* CFD code validation and benchmarking against BFBT boiling flow experiment
BOI-10	*L. Vyskocil, J. Macek* Boiling flow simulation in NEPTUNE_CFD and FLUENT codes
BOI-11 (Poster)	*J. Macek, L. Vyskocil* Simulation of critical heat flux experiments in NEPTUNE_CFD code

Technical Session MS

Multi-Scale Analysis

MS-01	*F. Cadinu, T. Kozlowski, P. Kudinov* Study of algorithmic requirements for a system-to-CFD coupling Strategy
MS-02	*D. Jamet, O. Lebaigue, C. Morel, and B. Arcen* Towards a multi-scale approach of two-phase flow modelling in the context of DNB modelling
MS-03	*D. Lakehal* LEIS for the prediction of turbulent multi-fluid flows with and without phase change applied to thermal-hydraulics
MS-04	*A. Dehbi* Assessment against DNS data of a coupled CFD-stochastic model for particle dispersion in turbulent channel flows

Technical Session CSG

Core and Steam Generators

CSG-01	*M. E. Conner, E. Baglietto, A.M. Elmahdi* CFD methodology and validation for single-phase flow in PWR fuel assemblies
CSG-02	*D. Tar, G. Baranyai, Gy. Ézsol, I. Tóth* Experimental investigation of coolant mixing in VVER reactor fuel bundles by particle image velocimetry
CSG-03 (Poster)	*K.S. Dolganov, A.V. Shishov* Cross-verification of one- and three-dimensional models for VVER steam generator
CSG-04 (Poster)	*T. Ikeno, S. Kakinoki* Experimental and numerical approach to validate pressure loss predictability of a commercial code
CSG-05 (Poster)	*V.F. Strizhov, M.A. Bykov, A.Ye. Kiselev .V. Shishov, A.A. Krutikov, D.A. Posysaev, D.A. Mustafina* Development of a 3D model of tube bundle of VVER reactor steam generator

Technical Session AR

Advanced Reactors

AR-01 *K. D. Hamman, R. A. Berry*
A CFD M&S Process for fast reactor fuel assemblies

AR-02 *I. Kei Ito, T. Kunugi,. H. Ohshima*
Development and validation of high-precision CFD method with Volume-Tracking algorithm for gas-liquid two-phase flow simulation on unstructured mesh

AR-03 *H. M. McIlroy, D. M. McEligot, R. J. Pink*
Idaho National Laboratory program to obtain benchmark data on the flow phenomena in a scaled model of a prismatic gas-cooled reactor lower plenum for the validation of CFD codes?

AR-04 (Poster) *N. Kimura, K. Hayashi, H. Kamide*
Experimental approach to flow field evaluation in upper plenum of reactor vessel for innovative sodium cooled fast reactor

AR-05 (Poster) *D.Ramdasu, N.S. Shivakumar, G. Padmakumar, C. Anand Babu, G. Vaidyanathan, S. Rammohan, S.K Sreekala, S. Manikandan, S. Saseendran*
Validation by Experiments for gas entrainment studies in 5/8 surge tank model of PFBR

Conclusions and Recommendations

There were over 140 participants to the XCFD4NRS workshop to hear 5 invited talks, 3 talks on OECD-CSNI activities related to CFD, 44 technical papers, and to see 15 posters. This is about a 40% increase with respect to the previous CFD4NRS held in Garching in 2006, and this confirms that there is a real need for such workshops. The original objective that 2/3 of the papers be concerned with two-phase issues and 1/3 dedicated to experimental techniques and CFD grade experimental data was achieved. Many participants sent the message that the workshop was well organised.

The USA is a candidate to host a follow-up meeting, organized by the US-NRC (confirmed by NRC a few days after the workshop). The suggestion received encouraging remarks from the audience during the discussion at the panel session. KAERI also proposed to host and organize a future workshop. The majority of participants considered they would be interested in attending a follow-up workshop within two years. Comments were made during the panel session on the content of XCFD4NRS. It was considered that some contributions were not directly related to the nuclear safety. Another comment suggested that such workshops should be a forum to discuss novel approaches, but that one must also keep in mind that the end users are people from the nuclear safety area. There was a consensus on the need to maintain the high quality of the papers. It was also suggested to promote international benchmarks for CFD.

Both the CFD4NRS and XCFD4NRS workshops proved to be a very valuable means to assess the status of CFD code capabilities and validation, to exchange experiences in CFD code applications, and to monitor progress. There was again an offer to publish selected papers from the workshop in a special issue of the Nuclear Engineering and Design (NED) journal. It was also mentioned that the special issue devoted to CFD4NRS received a very high number of visits on the journal website, and many of the papers have subsequently been downloaded. Session chairmen will make a selection of papers to be submitted to the NED Journal. It was anticipated that the special issue of NED dedicated to XCFD4NRS would appear in 2010.

The following additional comments were made:

- Current capabilities of two-phase measurement techniques are still too limitative for CFD validation. Further efforts are required to develop more advanced techniques, such as X-ray PIV, and international cooperation is necessary to support the high cost of development.
- Most of CFD codes are commercial and do not offer a full transparency with access to sources, which may be a problem from a regulation point of view.
- Most of CFD codes are commercial and do not offer a full transparency with access to sources, which may be a problem from a regulatory point of view.
- Application of CFD to Nuclear Safety requires that code uncertainties are determined, as they are now for system codes.

The participants made the following recommendations:

- One should keep a close link between people developing experimental techniques and performing validation experiments, and people developing CFD models and codes.
- Best Practice Guidelines should still be promoted, which requires that they are further developed and made more specific to each application. For two-phase CFD the establishment of Guidelines on the choice of the physical models depending on the phenomena being investigated has to be considered as a long-term activity.
- Experimental techniques should be further developed to provide CFD-grade data for validating CFD models, including estimates of measurement uncertainties.
- A new item should be added in the scope of the workshop: the development and application of uncertainty evaluation methods for CFD codes.

CFD4NRS-3: Experimental Validation and Application of CFD and CMFD Codes to Nuclear Reactor Safety Issues

Background

Computational methods have been used in the safety analysis of nuclear reactor systems for more than thirty years. During this time, reliable codes have been developed for analysing the primary system and the secondary system response, and similar programmes have also been written for containment and severe accident analyses. These codes are written as networks of 1-D or even 0-D cells. It is evident, however, that the flow in many reactor primary components is essentially 3-D in nature, as e.g. in natural circulation, and mixing and stratification in containments. Computational Fluid Dynamics (CFD) has the potential to treat flows of this type, and to handle geometries of almost arbitrary complexity. Hence, CFD is expected to feature more frequently in reactor thermal-hydraulics in the future, as over the last decade, three-dimensional CFD codes have been increasingly used to predict steady-state and transient flows in nuclear reactor safety (NRS) applications. The reason for the increased use of multidimensional CFD methods is not only the increased availability of capable computer systems but also the ongoing drive to improve and reduce uncertainty in our predictions of important phenomena, e.g., pressurized thermal shock, boron mixing, and thermal striping and to address new design features such as advanced accumulators and helical steam generators.

However, while traditional approaches to Nuclear Reactor Safety (NRS) analysis, using system codes for example, have been successful because a large database of mass, momentum and energy exchange correlations (from essentially 1-D special effect experiments) has been built to them, analogous data for 3-D flows is very sparse in comparison, making CFD codes for 3-D NRS applications limited. In fact, the main difficulty is that industrial-type CFD is highly non-linear, and resolution of flow structures spanning a wide range of scales (e.g. boundary and free-shear layers, vertical structures, zones of recirculation, etc.) is required. CFD codes contain empirical models for simulating turbulence, heat transfer, multiphase flows, and chemical reactions. Such models should be validated before they can be used with sufficient confidence in NRS applications. The necessary validation is performed by comparing model results against trustworthy data. A reliable model assessment requires CFD simulations with control of numerical errors to avoid erroneous conclusions being drawn concerning the performance of the physical models employed in the simulation. In addition, despite the increased availability of capable computer systems, challenges abound when one is faced with a requirement to simulate a full-scale reactor scenario.

Although reactor system code models will still play a key role in the future for full transient analyses, there will be critical safety issues requiring the resolution provided by advanced three dimensional CFD codes. With proposed design features, CFD will play an ever-increasing role in the safety analysis of future reactor designs. Currently, some safety authorities (e.g., NRC) and industry have started utilizing CFD codes for a better estimation of uncertainties and to improve the basis for regulatory and design decisions. It is therefore important that the nuclear community (research and safety authorities as well as the industry) spend time and resources to validate and demonstrate the applicability of CFD codes for various reactor safety issues. The mixing-T benchmark exercise presented in this workshop is a good example of these efforts.

All these issues have prompted an Organization for Economic Cooperation and Development/Nuclear Energy Agency (OECD/NEA) initiative to form writing groups of experts with the specific task of assessing the maturity of CFD codes for NRS applications and to establish a database and best practice guidelines for their validation and use. The CFD4NRS-3 Workshop is a development from these activities, and follows the two previous CFD4NRS workshops held in Garching, Germany (Sept. 2006) and Grenoble, France (Sept. 2008).

Scope

The purpose of the workshop was to provide a forum for numerical analysts and experimentalists to exchange information in the field of NRS-related activities relevant to CFD validation, with the objective of providing input to WGAMA CFD experts to create a practical, state-of-the-art, web-based assessment matrix on the use of CFD for NRS applications. The workshop included single-phase and multiphase CFD applications as well as new experimental techniques, including the following:

- Single-phase and two-phase CFD simulations with an emphasis on validation were sought in areas such as boiling flows, free-surface flows, direct contact condensation, and turbulent mixing. These should relate to NRS-relevant issues such as pressurized thermal shock, critical heat flux, pool heat exchangers, boron dilution, hydrogen distribution, and thermal striping. The use of systematic error quantification and Best Practice Guidelines (BPGs) was encouraged.
- Experiments providing data suitable for CFD validation — specifically in the area of NRS — including local measurement devices such as multi-sensor optical or electrical probes, Laser Doppler Velocimetry (LDV), hot-film/wire anemometry, Particle Image Velocimetry (PIV), Laser-Induced Fluorescence (LIF), and other innovative techniques. It was strongly recommended that the papers include a discussion of measurement uncertainties.

Welcoming Address

B. Sharon (US NRC, USA)

Invited Lectures

1. *J. H. Mahaffy (PSU, USA)*
   Synthesis of T-Junction Benchmark Results
2. *K. Okamoto (Univ. Tokyo, Japan)*
   Best Practice Procedures on Performing Two-Phase Flow Experiments for CFD Validation
3. *K. C. Mousseau (INL, USA)*
   Computational Fluid Dynamics and Experimental Fluid Dynamics Database

4. *O. Simonin (INPT, France)*
   CFD Modeling of Dispersed Two-Phase Flow
5. *E. Laurien (Univ. Stuttgart, Germany)*
   Numerical Simulation of Flow and Heat Transfer of Fluids at Supercritical Pressure

Technical Session 1

Advanced Reactors (1)

1. *M. Böttcher*
   CFD Analysis of Decay Heat Removal Scenarios of the Lead Cooled ELSY Reactor
2. *R. W. Johnson*
   Evaluation of an Experimental Data Set to Be Validation Data for CFD for a VHTR
3. *A. Onea, M. Böttcher, D. Struwe*
   Lead Pressure Loss in the Heat Exchanger of the ELSY Fast Lead-Cooled Reactor by CFD Approach
4. *U. Bieder, V. Barthel, F. Ducros, P. Quéméré, S. Vandroux*
   CFD Calculations of Wire Wrapped Fuel Bundles: Modelling and Validation Strategies

Technical Session 2

Containment (1)

5. *B. Schramm, J. Stewering, M. Sonnenkalb*
   Validation of a Simple Condensation Model for Simulation of Gas Distributions in Containments with CFX
6. *M.A. Mohaved, J.R. Travis*
   Assessment of the Gasflow Spray Model Based on the Calculations of the Tosqan Experiments 101 and 113
7. *T.J.H. Pättikangas, J. Niemi, J. Laine, M. Puustinen, H. Purhonen*
   CFD Modelling of Condensation of Vapour in the Pressurized Poolex Facility
8. *A. Zirkel, E. Laurien*
   Investigation of the Turbulent Mass Transport during the Mixing of a Stable Stratification with a Free Jet Using CFD Methods

Technical Session 3

Boiling Flow (1)

9. *I. Kataoka, K. Yoshida, M. Naitoh, H. Okada, T. Mori*
   Modeling of Turbulent Transport Term of Interfacial Area Concentration in Gas-Liquid Two-Phase Flow
10. *D. Bestion*
    Applicability of Two-Phase CFD to Nuclear Reactor Thermalhydraulics and Elaboration of Best Practice Guidelines
11. *P. Ruyer, K. Keshk, F. Deffayet, Ch. Morel, J. Pouvreau, F. François*
    Numerical Simulation of Condensation in Bubbly Flow
12. *A. Douce, S. Mimouni, M. Guingo, C. Morel, J. Laviéville, C. Baudry*
    Validation of Neptune_CFD 1.0.8 for Adiabatic Bubbly Flow and Boiling Flow

Technical Session 4

Bundle Flow

13. *E. Dominguez-Ontiveros, Y. A. Hassan, M. E. Conner, Z. Karoutas*
    Experimental Benchmark Data for PWR Rod Bundle with Spacer-Grids
14. *H. S. Kang, S. K. Chang, C.-H. Song*
    CFD Analysis of the Matis-H Experiments on the Turbulent Flow Structures in a 5x5 Rod Bundle with Mixing Devices
15. *J. Yan, K. Yuan, E. Tatli, D. Huegel, Z. Karoutas*
    CFD Prediction of Pressure Drop for the Inlet Region of a PWR Fuel Assembly
16. *E. Merzari, W.D. Pointer, J. G. Smith*
    Numerical Simulation of the Flow in Wire-Wrapped Pin Bundles: Effect of Pin-Wire Contact Modeling

Technical Session 5

Fire

17. *M.A. Mohaved*
    Recommendation for Maximum Allowable Mesh Size for Plant Combustion Analyses with CFD Codes
18. *C. Lapuerta, F. Babik, S. Suard, L. Rigollet*
    Validation Process of the Isis CFD Software for Fire Simulation
19. *H. S. Kang, S. B. Kim, M.-H. Kim, H. C. No*
    CFD Analysis of a Hydrogen Explosion Test with High Ignition Energy in Open Space

Technical Session 6

Dry Cask

20. *G. Banken, K. Tavassoli, J. Bondre*
    Validation of Computational Fluid Dynamics Code Models for Used Fuel Dry Storage Systems
21. *G. Zigh, J. Jolis, J. A. Fort*
    A 2-D Test Problem for CFD Modeling Heat Transfer in Spent Fuel Transfer Cask Neutron Shields
22. *E. Lindgren, S. Durbin*
    Pressure Drop Measurement of Laminar Air Flow in Prototypic BWR and PWR Fuel Assemblies
23. *K. Das, D. Basu, J. Solis, G. Zigh*
    Computational Fluid Dynamics Modeling Approach to Evaluate VSC-17 Dry Storage Cask Thermal Designs
24. *I. Rampall, K. K. Niyogi, D. Mitra-Majumdar*
    Validation of the FLUENT CFD Computer Program by Thermal Testing of a Full Scale Double-Walled Prototype Canister for Storing Chernobyl Spent Fuel

Technical Session 7

Advanced Reactors (2)

25. *S. B. Rodriguez, S. Domino, M. S. El-Genk*
    Fluid Flow and Heat Transfer Analysis of the VHTR Lower Plenum Using the Fuego CFD Code

26. *J.R. Buchanan, Jr., R.C. Bauer*
    Experimental Efforts for Predictive Computational Fluid Dynamics Validation
27. *A. Dehbi, S. Martin*
    Particle Deposition on an Array of Spheres Using RANS-RSM Coupled to a Lagrangian Random Walk
28. *B. Wilson, J. Harris, B. Smith, R. Spall*
    Unsteady Validation Metrics for CFD in a Cylinder Array

Technical Session 8

Boiling Flow (2)

29. *B.J. Yun, A. Splawski, S. Lo, C.-H. Song*
    Prediction of a Subcooled Boiling Flow with Mechanistic Wall Boiling and Bubble Size Models
30. *D. Prabhudharwadkar, M. Lopez de Bertodano, J. Buchanan Jr.*
    Assessment of the Heat Transfer Model and Turbulent Wall Functions for Two Fluid CFD Simulations of Subcooled and Saturated Boiling
31. *L. Vyskocil, J. Macek*
    CFD Simulation of Critical Heat Flux in a Tube
32. *C. Gerardi, H. Kim, J. Buongiorno*
    Use of Synchronized, Infrared Thermometry and High-Speed Video for Generation of Space- and Time-Resolved High-Quality Data on Boiling Heat Transfer

Technical Session 9

Mixing Flow (1)

33. *G. Pochet, M. Haedens, C.R. Schneidesch, D. Léonard*
    CFD Simulations of the Flow Mixing in the Lower Plenum of PWRs
34. *D. R. Shaver, S. P. Antal, M. Z. Podowski, D. H. Kim*
    Direct Steam Condensation Modeling for a Passive PWR Safety System
35. *B. Yamaji, R. Szijártó, A. Aszódi*
    Study of Thermal Stratification and Mixing Using PIV
36. *C. Boyd, K. Armstrong*
    Challenges for the Extension of Limited Experimental Data to Full-Scale Severe Accident Conditions Using CFD

Technical Session 10

Plant Applications

37. *T.J.H. Pättikangas, J. Niemi, V. Hovi, T. Toppila, T. Rämä*
    Three-Dimensional Porous Media Model of a Horizontal Steam Generator
38. *G. M. Cartland Glover, E. Krepper, H. Kryk, F.-P. Weiss, S. Renger, A. Seelinger, F. Zacharias, A. Kratzsch, S. Alt, W. Kästner*
    Fibre Agglomerate Transport in a Horizontal Flow
39. *P. Nilsson, E. Lillberg, N. Wikström*
    LES with Acoustics and FSI for Deforming Plates in Gas Flow
40. *L. Mengali, D. Melideo, F. Moretti, F. D'Auria, O. Mazzantini*
    CFD Calculation of the Pressure Drop through a Rupture Disk

41. *Y. S. Bang, G. S. Lee, S.-W. Woo*
A Shallow Water Equation Solver and Particle Tracking Method to Evaluate the Debris Transport

Technical Session 11

Pressurized Thermal Shock

42. *P. Apanasevich, D. Lucas, T. Höhne*
Pre-Test CFD Simulations on Topflow-PTS Experiments with ANSYS CFX 12.0
43. *M. Scheuerer, J. Weis*
Transient Computational Fluid Dynamics Analysis of Emergency Core Cooling Injection at Natural Circulation Conditions
44. *P. Coste, J. Laviéville, J. Pouvreau, C. Baudry, M. Guingo, A. Douce*
Validation of the Large Interface Method of Neptune_CFD 1.0.8 for Pressurized Thermal Shock (PTS) Applications
45. *M. Labois, D. Lakehal*
PTS Prediction Using the CMFD Code TransAT: the COSI Test Case

Technical Session 12

Containment (2)

46. *J. Stewering, B. Schramm, M. Sonnenkalb*
Validation of CFD-Models for Natural Convection, Heat Transfer and Turbulence Phenomena
47. *D. Paladino, M. Andreani, R. Zboray, J. Dreier*
Toward a CFD-Grade Database Addressing LWR Containment Phenomena
48. *E. Studer, J. Brinster, I. Tkatschenko, G. Mignot, D. Paladino, M. Andreani*
Interaction of a Light Gas Stratified Layer with an Air Jet Coming from Below: Large Scale Experiments and Scaling Issues
49. *J. Yáñez, A. Kotchourko, A. Lelyakin*
Hydrogen Deflagration Simulations under Typical Containment Conditions for Nuclear Safety

Technical Session 13

Boiling Flow (3)

50. *D. Lucas, M. Beyer, L. Szalinski*
Experimental Data on Steam Bubble Condensation in Poly-Dispersed Upward Vertical Pipe Flow
51. *J. L. Muñoz-Cobo, S. Chiva, S. Mendes, M. A. Abdelaziz*
Coupled Lagrangian and Eulerian Simulation of Bubbly Flows in Vertical Pipes: Validation with Experimental Data Using Multi-Sensor Conductivity Probes and Laser Doppler Anemometry
52. *C. Lifante, T. Frank, A.D. Burns, D. Lucas, E. Krepper*
Prediction of Polydisperse Steam Bubble Condensation in Sub-Cooled Water Using the Inhomogeneous Musig Model

Technical Session 14

Mixing Flow (2)

53. *J. Kim, J. J. Jeong*
    Large Eddy Simulation of a Turbulent Flow in a T-Junction
54. *M. Tanaka, H. Ohshima*
    Numerical Simulations of Thermal-Mixing in T-Junction Piping System Using Large Eddy Simulation Approach
55. *S.T. Jayaraju, E.M.J. Komen, E. Baglietto*
    Large Eddy Simulations for Thermal Fatigue Predictions in a T-Junction: Wall-Function or Wall-Resolved-Based LES
56. *V.M. Goloviznin, S.A. Karabasov, M.A. Zaitsev*
    Towards Empiricism-Free Large Eddy Simulation for Thermo-Hydraulic Problems
57. *R.B. Oza, V.D. Puranik, H.S. Kushwaha, K. Prasad, A. Murthy*
    Dispersion of Radionuclides and Radiological Dose Computation over a Mesoscale Domain Using Weather Forecast and CFD Model

Poster Session 2

1. *L. Vyskocil, J. Macek*
   CFD Simulation of Critical Heat Flux in a Rod Bundle
2. *R. Szijártó, B. Yamaji, A. Aszódi*
   Study of Natural Convection around a Vertical Heated Rod Using PIV/LIF Technique
3. *V.V. Chudanov, A.E. Aksenova, V.A. Perchiko, A.A. Makarevich, N.A. Pribaturin, O.N. Kashinskii*
   3D CFD Conv Code: Validation and Verification
4. *D. Melideo, F. Moretti, F. Terzuoli, F. D'Auria, O. Mazzantini*
   Calculation of Pressure Drops through Atucha-II Fuel Assembly Spacer Grids
5. *S. Durbin, E. Lindgren, A. Zigh*
   Measurement of Laminar Velocity Profiles in a Prototypic PWR Fuel Assembly
6. *S. Mimouni, N. Mechitoua, E. Moreau, M. Ouraou*
   CFD recombiner modelling and validation on the H2-Par and Kali-H2 experiments

Poster Session 3

7. *I.A. Bolotnov, F. Behafarid, D.R. Shaver, S.P. Antal, K.E. Jansen, R. Samulyak, H. Wei and M.Z. Podowski*
   Multiscale Computer Simulation of Fission Gas Discharge During Loss-of-Flow Accident in Sodium Fast Reactor
8. *A. Foissac, J. Malet, R.M. Vetrano, J.-M. Buchlin, S. Mimouni, F. Feuillebois, O. Simonin*
   Experimental Measurements of Droplet Size and Velocity Distributions at the Outlet of a Pressurized Water Reactor Containment Swirling Spray Nozzle
9. *S. Mimouni, N. Mechitoua, A. Foissac, M. Hassanaly, M. Ouraou*
   CFD Modeling of Wall Steam Condensation: Two Phase Flow Approach Versus Homogeneous Flow Approach
10. *A. Tentner, S. Lo, D. Pointer, A. Splawski*
    Advances in the development and validation of CFD- BWR, a Two-Phase Computational Fluid Dynamics Model for the Simulation of Flow and Heat Transfer in Boiling Water Reactors

Poster Session 4

11. *G. Tryggvason, J. Buongiorno*
    The Role of Direct Simulations in Validation and Verification

12. *J.A. Dixon, A. Guijarro Valencia, P. Ireland, P. Ridland, N. Hills*
    A Coupled CFD Finite Element Analysis Methodology in a Bifurcation Pipe in a Nuclear Plant Heat Exchanger
13. *K. Myllymäki, T. Toppila, T. Brandt*
    Interpreting Thermocouple Reading in Fuel Assembly Head – A CFD Studyy on Coolant Mixing
14. *H. Li, P. Kudinov*
    Effective Approaches to Simulation of Thermal Stratification and Mixing in a Pressure Suppression Pool
15. *C.-T. Tran, P. Kudinov*
    A Synergistic Use of CFD, Experiments and Effective Convectivity Model to Reduce Uncertainty in BWR Severe Accident Analysis

Poster Session 5

16. *D. Soussan, S. Pascal Ribot, M. Grandotto*
    2D Simulation of Two-Phase Flow across a Tube Bundle with Neptune_CFD Code
17. *N. Mechitoua, S. Mimouni, M. Ouraou, E. Moreau*
    CFD Modelling of the Test 25 of the Panda Experiment
18. *M. Labois, J. Panyasantisuk, T. Höhne, S. Kliem, D. Lakehal*
    On the Prediction of Boron Dilution Using the CMFD Code Transat: the Rocom Test Case

Conclusions and Recommendations

There were over 200 registered participants at the CFD4NRS-3 workshop. The program consisted of about 75 technical papers. Of these, 57 were oral presentations and 18 were posters. An additional 20 posters related to the OECD/NEA–sponsored CFD benchmark exercise on thermal fatigue in a T-Junction were presented. In addition, 5 keynote lectures were given by distinguished experts. This is about a 30% increase with respect to the previous XCFD4NRS workshop held in Grenoble in 2008, and a 70% increase compared to the first CFD4NRS workshop held in Garching in 2006, confirming that there is a real and growing need for such workshops.

The papers presented in the conference tackled different topics related to nuclear reactor safety issues. The conference consisted of 14 technical sessions. Among the topics included were containment, advanced reactors, multiphase flows, flow in a rod bundle, fire analysis, flows in dry casks, thermal analysis, mixing flows and pressurized thermal shock (PTS). About 1/3 of the papers were concerned with two-phase flow issues and the rest were devoted to single-phase CFD validation.

South Korea is a candidate to host a follow-up meeting scheduled in 2012, organized by KAERI. KAERI also volunteered to sponsor and organize the second OECD/NEA CFD benchmark exercise. In the closure meeting after the panel session discussion, the representative from the Paul Scherrer Institut (PSI) proposed to host a future workshop scheduled for 2014, and to organize and sponsor the third OECD/NEA benchmark exercise based on a stratification experiment in the PANDA facility at PSI. The great majority of participants were interested in attending a follow-up workshop within two years.

Comments were made during the panel session on the content of CFD4NRS-3. Two of the comments are that experiments can provide insight into the physics, and that CFD is now an accepted analysis tool, though it is very important to follow BPGs. There was a consensus on the need to maintain the high quality of the papers. The promotion of international benchmarking exercises for CFD was strongly encouraged. Another comment suggested that such workshops should be a forum to discuss novel approaches, but that one must also keep in mind that the end users are people from the nuclear safety community. The CFD4NRS, XCFD4NRS and CFD4NRS-3 workshops have proved to be very valuable means to assess the status of CFD code capabilities and validation, to exchange experiences in CFD code applications, and to monitor future progress.

There was again an offer to publish selected papers from the workshop in a special issue of the Nuclear Engineering and Design (NED) journal. It was also mentioned that the special issue devoted to CFD4NRS and XCFD4NRS received a very high number of visits on the journal website and a large number of papers were subsequently downloaded. Session chairmen will make a selection of papers to be submitted to the NED Journal. It is anticipated that the special issue of NED dedicated to CFD4NRS-3 will appear early in 2012.

The following additional comments were made:

- Collaboration between academia and industry is occurring and producing valuable results.
- It is useful to keep a view of the physics when interpreting the adequacy of CFD predictions.
- Challenges abound when one is faced with a requirement to simulate a full-scale reactor scenario, because there is often little relevant experimental data, there is often uncertainty in the boundary conditions, and that the need for grid sensitivity studies must be balanced against computational resources.
- When applying CFD to real problems, one should never lose sight of the overall picture in order to guide the decision-making in respect to the details of the CFD modelling approach.
- Current capabilities of two-phase measurement techniques are still too limited for CFD validation.
- Further efforts are required to develop more advanced techniques, such as X-ray PIV, and international cooperation is necessary to support the high cost of model development.
- Many CFD codes are commercial in origin and do not offer full transparency in respect to access to source code, which may be a problem from a regulatory point of view.
- Application of CFD to NRS issues requires that code uncertainties be determined, as they are now for system codes.

The participants made the following recommendations:

- One should keep a close link between people developing experimental techniques and performing validation experiments, and the people developing CFD models and codes.
- There is still limited use of BPGs in many applications, and often there is use of only one computational grid, sometimes even with first-order spatial discretization. This clearly limits understanding, since the physical and numerical errors are still superimposed.
- Best Practice Guidelines should still be promoted, which requires that they are further developed and made more application-specific. For two-phase CFD, the establishment of guidelines on the choice of the physical models depending on the phenomena being investigated has to be considered as a longterm activity.
- The papers indicated a consideration of CFD best practice guidelines, but their use is not documented in a systematic way by the authors.
- The presentations in the workshop demonstrated virtually universal awareness and attention to BPGs, but with varied success in practical implementation.
- A good application of CFD doesn't necessarily provide "margin", but helps to understand its physical justification when such margin exists.
- Experimental techniques should be further developed to provide CFD-grade data for validating CFD models, including estimates of measurement uncertainties.

- It appears that CFD is now state-of-the-art for computing adiabatic bubbly flows, and that the implementation of heat and mass transfer models for boiling and condensation has begun. One can also expect advancements in the use of CFD to study boiling and condensation at a fundamental level in the near future.

CFD4NRS-4

The Experimental Validation and Application of CFD and CMFD Codes to Nuclear Reactor Technology

Background

The last decade has seen an increasing use of three-dimensional CFD and CMFD codes in predicting single-phase and multi-phase flows under steady-state or transient conditions in nuclear reactors. The reason for the increased use of multi-dimensional CFD methods is that a number of important thermal-hydraulic phenomena cannot be predicted using traditional one-dimensional system analysis codes with the required accuracy and spatial resolution. CFD codes contain empirical models for simulating turbulence, heat transfer, multi-phase interaction and chemical reactions. Such models must be validated before they can be used with sufficient confidence in nuclear reactor safety (NRS) applications.

The necessary validation is performed by comparing model predictions against trustworthy data. However, reliable model assessment requires CFD simulations to be undertaken with full control over numerical errors and input uncertainties to avoid erroneous conclusions being drawn. These requirements have prompted an OECD/NEA initiative to form writing groups of experts with the specific task of assessing the maturity of CFD codes for NRS applications, and to establish a data base and Best Practice Guidelines (BPGs) for their validation.

Scope

Following the CFD4NRS workshops held in Garching, Germany (Sept. 2006), Grenoble, France (Sep. 2008) and Washington D.C., USA (Sept. 2010), this Workshop is intended to extend the forum created for numerical analysts and experimentalists to exchange information in the application of Computational Fluid Dynamics (CFD) and Computational Multi-Fluid Dynamics (CMFD) to nuclear reactor safety issues. The workshop includes single-phase and multi-phase CFD applications, and offers the opportunity to present new experimental data for CFD validation. Emphasis has been in the following areas:

- More emphasis has to be given on the experiments, especially on two-phase flow, for advanced CMFD modeling for which sophisticated measurement techniques are required.
- It is very important to deepen understanding the physics before numerical analysis.
- Single-phase and multi-phase CFD simulations with a focus on validation are welcome in areas such as: single-phase heat transfer, boiling flows, free-surface flows, direct contact condensation and turbulent mixing. These should relate to NRS-relevant issues, such as pressurized thermal shock, critical heat flux, pool heat exchangers, boron dilution, hydrogen distribution in containments, thermal striping, etc. The use of systematic error quantification and the application of BPGs are strongly encouraged.
- Experiments providing data suitable for CFD or CMFD validation are also welcome. These should include local measurements using multi-sensor probes, laser-based techniques (LDV, PIV or LIF), hot-film/wire anemometry, imaging, or other advanced measuring techniques. Papers should include a discussion of measurement uncertainties.

Welcoming Address

W.-P. Baek (KAERI)

Invited Lectures

1. *D. Bestion (CEA, France)*
   The Difficult Challenge of a Two-Phase CFD Modelling for All Flow Regimes
2. *C.-H. Song (KAERI)*
   Synthesis of OECD/NEA-KAERI Rod Bundle Benchmark Exercise
3. *Richard R. Schultz (INL, USA)*
   Using CFD to Analyze Nuclear Systems Behaviour: Defining the Validation Requirements
4. *S. J. Lee (POSTECH, S. Kore)*
   Advanced Flow Visualization Technique for CFD Validation
5. *K. Ikeda (MHI, Japan)*
   CFD Application to Advanced Design for High Efficiency Spacer Grid

Technical Session 1

Advanced Reactors

1. *B.-U. Bae, S. Kim, Y.-S. Park, B.-D. Kim, K.-H. Kang*
   Multi-dimensional temperature distribution in PCCT (Passive Condensation Cooling Tank) and PCHX (Passive Condensation Heat Exchanger) of PAFS (Passive Auxiliary Feedwater System)
2. *M. Tanaka*
   Uncertainty Quantification Scheme in V&V of Fluid-Structure Thermal Interaction Code for Thermal Fatigue Issue in a Sodium-cooled Fast Reactor
3. *Y. Xu, J. Yan, K. Yuan, C. Fu, P. Xu, S. Ray*
   CFD Multi-Physics Analysis of Fuel Bundles under Accidental Conditions for New Fuel Designs

Technical Session 2

Condensation

4. *P. Coste, A. Ortolan*

Two-Phase CFD PTS Validation in an Extended Range of Thermo Hydraulics Conditions Covered by the COSI Experiment

5. *A. Dehbi, F. Janasz, B. Bell*
   Validation of a CFD Model for Steam Condensation in the Presence of Non-condensable Gases
6. *L. Vyskocil, J. Schmid, J. Macek*
   CFD Simulation of Air-Steam Flow with Condensation
7. *G. Zschaeck, T. Frank, A. D. Burns*
   CFD Modelling and Validation of Wall Condensation in the Presence of Non-condensable Gases

Technical Session 3

Boiling/Bubbly Flow (1)

8. *K. Fu, H. Anglart*
   Implementation and Validation of Two-Phase Boiling Flow Models in OpenFOAM
9. *J. Peltola, T.J.H. Pättikangas*
   Development and Validation of a Boiling Model for OpenFOAM Multiphase Solver
10. *E. Krepper, R. Rzehak, C. Lifante, Th. Frank*
    CFD for Subcooled Flow Boiling: Coupling Wall Boiling and Population Balance Models
11. *Y. Liao, D. Lucas, E. Krepper*
    Application of New Closure Models for Bubble Coalescence and Breakup to Steam-Water Pipe Flow

Technical Session 4

Bundle Flow (1)

12. *S.-K. Chang, S. Kim, C.-H. Song*
    OECD/NEA – MATiS-H Rod Bundle CFD Benchmark Exercise Test
13. *U. Bieder*
    Analysis of the Flow Down and Upwind of Split-Type Mixing Vanes
14. *Th. Frank, S. Jain, A.A. Matyushenko, A.V. Garbaruk*
    The OECD/NEA MATiS-H Benchmark – CFD Analysis of Water Flow through a 5x5 Rod Bundle with Spacer Grids using ANSYS Fluent and ANSYS CFX

Technical Session 5

Bundle Flow (2)

15. *A. Kiss, A. Aszódi*
    Sensitivity Studies on CFD Analysis for Heat Transfer of Supercritical Water Flowing in Vertical Tubes
16. *J. Yan, M. E. Conner, R. A. Brewster, Z. E. Karoutas, E. E. Dominguez-Ontiveros, Y. A. Hassan*
    Validation of CFD Method in Predicting Steady and Transient Flow Field Generated by PWR Mixing Vane Grid
17. *Y .V. Yudov*
    Using the DINUS Code for Direct Numerical Simulation of Hydrodynamic Processes in VVER-440 Fuel Rod Bundles

Technical Session 6

Hydrogen Transport and Fire

18. *H. S. Kang, S. B. Kim, M.-H. Kim, H. C. No*
    CFD Analysis of a Hypothetical H2 Explosion Accident between the HTTR and the H2 Production Facility in JAEA
19. *S. Kelm, W. Jahn, E.-A. Reinecke, H.-J. Allelein*
    Passive Auto-Catalytic Recombiner Operation Validation of a CFD-approach against OECD-THAI HR2-test
20. *V. Shukla, P. Sivagangakumar, S. Ganju1, A. Kumar K. R. S. G. Markandeya*
    Development of CFD Based Numerical Tool for Addressing Hydrogen Transport and Mitigation Issues in the Containment of Nuclear Power Plants
21. *S. Worapittayaporn, L. Rudolph*
    Validation of Coupled BVM-EDM Combustion Model in ANSYS CFX for Hydrogen Combustion Calculation during Postulated Severe Accidents in NPP

Technical Session 7

Multi-scale & Multi-physics Analysis

22. *S. Haensch, D. Lucas, E. Krepper, T. Höhne*
    A CMFD-model for Multi-scale Interfacial Structures
23. *L. Vyskocil, J. Macek*
    Coupling of CFD Code with System Code and Neutron Kinetics Code
24. *M. Jeltsolv, K. Kööp, P. Kudinov, W. Villanueva*
    Development of Domain Overlapping STH/CFD Coupling Approach for Analysis of Heavy Liquid Metal Thermal Hydraulics in TALL-3D Experiment
25. *B. Gaudron, S. Jayaraju, S. Bellet, P. Freydier, D. Alvarez*
    Code_Saturne Integral Validation on ROCOM Test for Heterogeneous Inherent Boron Dilution Transient

Technical Session 8

Plant Applications (1)

26. *J. Bakosi, N. Barnett, M. A. Christon, M. M. Francois, R. B. Lowrie*
    Large-scale Turbulent Simulations of Grid-to-rod Fretting
27. *D. Melideo, F. Moretti, F. Terzuoli, F. D'Auria, O. Mazzantini*
    Optimization of the Atucha-II Fuel Assembly Spacer Grids
28. *D. Melideo, L. Mengali, F. Moretti, W. Giannotti, F. Terzuoli, F. D'Auria, O. Mazzantini*
    Development of a CFD Model for Investigation of Atucha-II Containment
29. *S.-G. Yang, E.-J. Park*
    CFD Simulations for APR+ Reactor Design

Technical Session 9

Bundle Flow (3)

30. *F. Barthel, R. Franz, E. Krepper, U. Hampel*
    Experimental Studies on Sub-cooled Boiling in a 3x3 Rod Bundle
31. *E. Dominguez-Ontiveros, Y. Hassan, R. Franz, R. Barthel, U. Hampel*
    Experimental study of a Simplified 3 X 3 Rod Bundle using DPIV
32. *C. Lifante, B. Krull, Th. Frank, R. Franz, U. Hampel*
    3x3 Rod Bundle Investigations. Part II: CFD Single-Phase Numerical Simulations

Technical Session 10

Plant Applications (2)

33. *C. Boyd, R. Skarda*
    CFD Predictions of Standby Liquid Control System Mixing in Generic BWR
34. *T. Hoehne, A. Grahn, S. Kliem*
    Numerical Simulation of the Insulation Material Transport to a Pressurized Water Reactor Core under Loss of Coolant Accident Conditions
35. *T. Rämä, T. Toppila, T. Kelavirta,P. Martin*
    CFD Analysis of the Temperature Field in Emergency Pump in LOVIISA NPP
36. *M. Ishigaki, T. Watanabe, H. Nakamura*
    Numerical Simulation of Two-Phase Critical Flow in a Convergent-divergent Nozzle

Technical Session 11

Boiling/Bubbly Flow (2)

37. *I.-C. Chu, H. C. No, C.-H. Song*
    Visualization of High Heat Flux Boiling and CHF Phenomena in a Horizontal Pool of Saturated Water
38. *D. Lucas, M. Banowski, D. Hoppe, M. Beyer, L. Szalinski, F. Barthel, U. Hampel*
    Experimental Data on Vertical Air-Water Pipe Flow Obtained by Ultrafast Electron Beam X-Ray Tomography Measurements
39. *R. Sugrue, T. McKrell, J. Buongiorno*
    On the Effects of Orientation Angle, Subcooling, Mass Flux, Heat Flux, and Pressure on Bubble Departure Diameter in Subcooled Flow Boiling
40. *G.H. Yeoh, S.C.P. Cheung, J.Y. Tu, D. Lucas, E. Krepper*
    Validation of Models for Bubbly Flows and Cap Flows using One-Group and Two-Group Average Bubble Number Density

Technical Session 12

Mixing

41. *F. Moretti, F. D'Auria*
    Addressing the Accuracy Quantification issue for CFD Investigation of In-Vessel Flows
42. *M. Gritskevich, A. V. Garbaruk, F. R. Menter*
    Investigation of the Thermal Mixing in a T-Junction Flow with Different SRS Approaches
43. *D. Kloeren, M. Kuschewski, E. Laurien*
    Large-Eddy Simulations of Stratified Flows in Pipe Configurations Influenced by a Weld Seam
44. *J. Xiong, X. Pan, S. Koshizuka, L. Zhang, X. Cheng*
    CFD Analysis on Localized Mass Transfer Enhancement in the Downstream of an Orifice

Poster Session 1

1. *M. A. Zaitsev, V. M. Goloviznin, S. A. Karabasov*
   A Highly Scalable Hybrid Mesh Cabaret Miles Method for MATIS-H Problem
2. *L. A. Golibrodo, N. A. Strebnev, M. M. Kurnosov, I. U. Galkin, I. K. Vdovkina*
   CFD Simulation of Turbulent Flow Structure in a Rod Bundle Array with the Split-Type Spacer Grid
3. *A. Batta, A. G. Class*
   CFD (Computational Fluid Dynamics) Study of Isothermal Water Flow in Rod Bundles with Split-type Spacer Grids: OECD/NEA Benchmark, MATiS-H

4. *N. Cinosi, S. Walker, M. Bluck, R. Issa, G. Hewitt*
   MATIS-H benchmark exercise with code STAR-CCM
5. *D. Chang, S. Tavoularis*
   Hybrid URANS/LES simulations of isothermal water flow in the MATiS-H rod bundle with a split-vane spacer grid
6. *A. Obabko, P. Fischer, E. Merzari, W. D. Pointer, T. Tautges*
   A Comparison of ID-DES and LES results for MATiS-H Benchmark
7. *A. Rashkovan, D. Novog*
   Turbulence Modeling Sensitivity Study for 2x2 and 5x5 Fuel Bundle
8. *L. Capone, S. Benhamadouche*
   MATiS-H benchmark. McMaster University contribution
9. *H. S. Kang, S. K. Chang, C.-H. Song*
   CFD Analysis of the OECD/NEA-KAERI Rod Bundle Benchmark Exercise with a Split Vane by RANS Turbulent Models of START-CCM+ 6.06

Poster Session 2

10. *H. Kwon, S. J. Kim, K. W. Seo, D. H. Hwang*
    Computations of Transient Natural Circulation on PNL 2 by 2 Test Bundle Experiments
11. *S. Kim, D. E. Kim, C. H. Song*
    Experimental Study on the Thermal Stratification and Natural Circulation Flow inside a Pool
12. *A. Nakamura, Y. Utanohara, K. Miyoshi, N. Kasahara*
    Simulation of Thermal Stripping at T-Junction Pipe Using LES with Mode Parameters and Temperature Diffusion Schemes
13. *S. J. Lee, H. K. Cho, K. H. Kang, S. Kim, H. Y. Yoon*
    Numerical Analysis of the Passive Condensation Cooling Tank (PCCT) using the CUPID Code

Video Session

1. *T. Yasui, S. Someya, K. Okamoto*
   Boiling Behavior of Droplets Impinging on Heated Liquid Metal Surface
2. *A. Ylönen, H.-M. Prasser*
   Cross-mixing in a Fuel Rod Bundle, Enhanced by Functional Spacer Grids Portraits of Liquid Film Flows
3. *M. Damsohn, D. Ito, R. Zboray, H.-M. Prasser*
   Portraits of Liquid Film Flows
4. *H.-M. Prasser*
   The Best of Wire-mesh Sensors -Inspirations for Their Future Use
5. *B. Niceno, Y. Sato*
   Numerical Modeling of Pool and Flow Boiling
6. *A.A. Matyushenko, A.V. Garbaruk, S. Jain, T. Frank*
   ANSYS Fluent results for the split type spacer grid geometry of the OECD/NEA MATiS-H Benchmark

Conclusions and Recommendations

There were over 150 registered participants at the CFD4NRS-4 workshop. The programme consisted of about 48 technical papers. Of these, 44 were presented orally and 4 as posters. An additional 8 posters related to the OECD/NEA–KAERI sponsored CFD benchmark exercise on turbulent mixing in a rod bundle with spacers (MATiS-H) were presented and a special session was allocated for 6 video presentations. In addition, five keynote lectures were given by distinguished experts.

The number of participants represents a 25% decrease with respect to the previous CFD4NRS-3 Workshop held in Washington DC in Septemeber 2010. Nonetheless, this attendance record compared favourably with the second Workshop in the series, XCFD4NRS, held in Grenoble in 2008, and a two-fold increase compared to the first Workshop, held in Garching in 2006. Factors influencing the slight fall in attendance are: (i) fewer domestic students; (ii) the NUTHOS-9 conference being held in Taiwan at exactly the same time; (iii) the expense involved in making the trip to Korea from Europe and (especially) the US; (iv) the negative impact on nuclear research following the Fukushima disaster in March 2011.

The papers given at the Workshop covered different nuclear safety topics, and, for the first time, some reactor design issues. However, the ratio of papers devoted to experimentation to those devoted to analysis was not as well balanced as previously seen, with too few experimental works reported. Progress in modelling, and improvements in the use of the Best Practice Guidelines for performing quality CFD computations can only result from pursuing a programme of analysis of a multitude of CFD-grade experiments. A wrong idea circulates, particularly among managers, that CFD simulations may ultimately replace costly experimentation. This is only partially true in the case of prototypic experiments, but CFD tools include many models and closure laws: these have to be properly validated, and this can only be achieved by means of experiments. It remains a primary objective of the CFD4NRS series of Workshops to bring together the experimenters providing the data needed to improve the physical models in CFD codes, and the analysts who utilise these models.

Switzerland is a candidate to host the next Workshop in 2014, and will be organised by staff at the Paul Scherrer Institute (PSI), who have also volunteered to sponsor and organise the third OECD/NEA CFD benchmark exercise, based on an experiment to be performed in the containment test facility PANDA. In the panel session at the close of the Workshop, delegates confirmed their interest in attending a follow-up Workshop, and considered the two-year interval to be appropriate.

As is customary at the panel session, which in this case was led by B. L. Smith (PSI) and D. Bestion (CEA), summaries were made by the respective session chairpersons of the presentations that were given during the 12 oral sessions, and comments invited from the audience. To open the session, A. Ulses (IAEA) expressed satisfaction with the organisation and smooth-running of the Workshop, and complimented the staff at KAERI on their efforts in this regard. The level of attendance confirmed the international level of interest in the theme and objectives of the Workshop, and he pledged continuing IAEA support for the future.

The session topics were wide and various, including advanced reactor modelling, flow mixing issues, boiling and condensation modelling, multiphase and multiphysics problems, containment analysis, plant application, hydrogen transport and fires, advanced measuring techniques, and single and multiphase flow in rod bundles. Comments arising from the summaries included:

- The nuclear CFD community should be encouraged to apply and further develop Uncertainty Qualification (UQ) methods in regard to their simulations, including uncertainties arising from the numerical solution procedure, the physical models employed, and in the initial and boundary conditions.
- Delegates appeared satisfied that the subject areas covered by the Workshop were comprehensive within the nuclear CFD community, and that leading experts in the field adequately covered the present state-of-the-art or projected future trends, as appropriate.
- It was noted that CFD is no substitute for properly understanding the basic thermal-hydraulic phenomena involved in the particular numerical analysis being undertaken. The CFD tools should be used instead to quantify the complex interplay between the various physical processes taking place.

- The current format, length and interval between CFD4NRS Workshops were generally considered appropriate, as was the rotation of venues worldwide. Hence no changes were proposed.
- The formula of combining the blind CFD benchmark activity with the occasion of the Workshop was appreciated, giving participants the possibility to display their work (as posters without accompanying papers), discuss their experiences with other participants, and visit the test facility on which the exercise was based. This practice will therefore be continued as far as possible in the future.
- Considerable interest was raised in the proposed forthcoming CFD benchmark on containment modelling and analysis, and to link the activity with CFD4NRS-5, giving people the opportunity to visit the PANDA facility.
- There was general appreciation of the local Workshop organisation (by KAERI staff), with only a few minor mishaps being voiced in regard to the arrangements made.
- All appreciated the open forum discussions that could take place during coffee breaks, the organised lunches and the conference banquet.
- Some concerns were raised that the quality of the papers was not as high as in previous Workshops in the series, and the panel chairman, on behalf of the organising committee, promised to address this issue seriously ahead of CFD4NRS-5.
- The analytical presentations at the Workshop demonstrated the almost universal application of Best Practice Guidelines in producing CFD simulations, including the use of higher order differencing methods for the fundamental equations. However, in reactor applications, the need for grid sensitivity studies still has to be balanced against computational resources.
- A similar code of practice in conducting experiments appears not to be so widespread, but the need for test data to be accompanied by error bars as a guide to measurement uncertainty is still to be encouraged for code validation tests.
- Several presentations showed that CFD was being used to guide the design of experiments in several key areas, and in the placement of instrumentation. This is a very welcome development.

APPENDIX 2: GLOSSARY

General

ADS	Automatic Depressurisation System (or Accelerator-Driven System)
AIAA	American Institute of Aeronautics and Astronautics
ANS	American Nuclear Society
APRM	Average Power Range Monitor
APWR	Advanced Pressurised Water Reactor
ASCHLIM	Assessment of Computational Fluid Dynamics Codes for Heavy Liquid Metals (EU 5th Framework Accompanying Measure)
ASME	American Society of Mechanical Engineers
ASTAR	Advanced Three-Dimensional Two-Phase Flow Simulation Tool for Application to Reactor Safety (EU 5th Framework Programme)
BDBA	Beyond Design-Basis Accident
BPGs	Best Practice Guidelines
CFD	Computational Fluid Dynamics
CMT	Core Make-up Tank
CPU	Central Processing Unit
CSNI	Committee on the Safety of Nuclear Installations
DBA	Design-Basis Accident
DES	Detached Eddy Simulation
DHX	Dumped Heat Exchanger
DNB	Departure from Nucleate Boiling
DNS	Direct Numerical Simulation
DRACS	Direct Reactor Auxiliary Cooling System
DVI	Direct Vessel Injection
ECCOMAS	European Community on Computational Methods in Applied Sciences
ECCS	Emergency Core-Cooling System
ECORA	Evaluation of Computational Fluid Dynamic Methods for Reactor Safety Analysis (EU 5th Framework Programme)
EOC	End-Of-Cycle
ERCOFTAC	European Research Community on Flow, Turbulence and Combustion
EUBORA	Boron Dilution Experiments (EU 4th Framework Concerted Action)
FISA-2003	The Fifth International Symposium on EU Research and Reactor Safety
FLOWMIX-R	Fluid Mixing and Flow Distribution in the Reactor Circuit (EU 5th Framework Shared-Cost Action)

GAMA	Working Group on the Analysis and Management of Accidents
HDC	Hydrogen Distribution and Combustion
HTC	Heat Transfer Coefficient
HPI	High Pressure Injection
HYCOM	Integral Large Scale Experiments on Hydrogen Combustion for Severe Accident Code Validation (EU 5th Framework Project)
IAEA	International Atomic Energy Agency
ICAS	International Comparative Assessment Study
IPSS	Innovative Passive Safety Systems (EU 4th Framework Programme)
IRWST	In-Containment Refuelling Water Storage Tank
ISP	International Standard Problem
JNC	Japanese Nuclear Corporation
JSME	Japanese Society of Mechanical Engineers
LANL	Los Alamos National Laboratory
LBLOCA	Large-Break Loss Of Coolant Accident
LES	Large Eddy Simulation
LFWH	Loss of Feedwater Heating
LOCA	Loss Of Coolant Accident
LPIS	Low Pressure Injection System
LPRM	Local Power Range Monitor
LS	Level Set
MCPR	Minimum Critical Power Ratio
NEA	Nuclear Energy Agency
NRS	Nuclear Reactor Safety
OECD	Organisation for Economic Cooperation and Development
PAHR	Post Accident Heat Removal
PRHR	Passive Residual Heat Removal
PIRT	Phenomena Identification Ranking Table
PTS	Pressurised Thermal Shock
RANS	Reynolds-Averaged Navier-Stokes
RPT	Recirculation Pump Trip
RPV	Reactor Pressure Vessel
RSM	Reynolds-Stress Model
SARA	Severe Accident Recriticality Analysis
SG	Steam Generator
SLB	Steam-Line Break
SM	Structure Mechanics
TEMPEST	Testing and Enhanced Modelling of Passive Evolutionary Systems Technology for containment cooling (EU 5th Framework Programme)
V&V	Verification and Validation
VOF	Volume-Of-Fluid
VTT	Technical Research Centre of Finland

Codes

ABAQUS	Commercial structural analysis program
AQUA	In-house CFD code developed by JNC
ANSYS	Commercial structural analysis program
APROS	In-house thermal-hydraulic code, developed Technical Research Centre of Finland
ASTEC	Accident Source Term Evaluation Code, developed jointly by IPSN and GRS for analysis of severe accidents
ATHLET	System analysis code, used extensively in Germany
CAST3M	General-purpose finite element code, developed by CEA
CATHARE	System analysis code, used extensively in France
ANSYS-CFX	Commercial CFD software program
COCOSYS	Containment code, developed by GRS for severe accident analysis
CONTAIN	Lumped-parameter code, sponsored by the US NRC, for severe accident analysis
DINUS-3	Direct Numerical Simulation (DNS) tool, developed by JNC
FELIOUS	Structural analysis code, developed by NUPEC
FLICA4	3-D, two-phase thermal-hydraulic code, developed by CEA/IPSN
FLUBOX	In-house, two-phase flow code, developed by GRS
FLUENT	Commercial CFD software program
GASFLOW	In-house CFD code developed by FZK
GENFLO	In-house CFD code, developed by VTT
GOTHIC	General-purpose containment code with 3-D capability, developed by Numerical Application Incorporated (NAI)
MCNP	Monte-Carlo Neutronics Program
MELCOR	Lumped-parameter code for analysing severe accidents, developed at Sandia NL
MpCCI	Mesh-based parallel Code Coupling Interface, distributed by STAR-CD/Adapco, used to couple CFD and SM codes
Permas	Commercial finite-element SM program
PHEONICS	Commercial CFD software program
RECRIT	Computer code for BWR recriticality and reflooding analyses, developed by VTT
RELAP5	System analysis code, used extensively in US and elsewhere
SAS4A	Sub-channel code, developed by ANL, used for analysis of severe accidents in liquid-metal-cooled reactors
SATURNE	3D CFD code, developed by EDF
SCDAP	Severe Core Damage Analysis Package, developed at Idaho National Laboratory
STAR-CD	Commercial CFD software program
TONUS	Containment code, developed by CEA under sponsorship of IRSN
TRAC	Transient Reactor Analysis Code
TRACE	TRAC/RELAP Combined Computational Engine
TRIO-U	CFD software program, developed by CEA
VSOP	Code for reactor physics and fuel cycle simulation, developed at FZJ

Experiments

MICOCO	Mixed Convection and Condensation benchmark exercise, based on MISTRA data
MISTRA	Experimental facility operated by CEA Saclay, used for containment studies
MSRE	Molten Salt Reactor Experiment, operated by ORNL
NOKO	Experimental facility at FZJ, used for studies of BWR condensers
PANDA	Integral test facility at PSI for analysis containment transients
PHEBUS	Experimental facility at CEA Cadarache, used for severe accident research
ROCOM	Experimental facility at FZR, used to investigate upper plenum mixing
RUT	Large-scale combustion experimental facility at the Kurchatov Institute, Russia
SETH	Series of experiments, sponsored by OECD, to be performed in the PANDA facility at PSI
UPTF	Upper Plenum Test Facility at FZR, examining LOCA-related phenomena

Reactors

ABWR	Advanced Boiling Water Reactor
ADS	Accelerator-Driven System
BWR	Boiling Water Reactor
EPR	European Pressurised-Water Reactor
ESBWR	European Simplified Boiling Water Reactor
GCR	Gas-Cooled Reactor
GFR	Gas-Cooled Fast Reactor
HDR	*Heissdampfreaktor;* reactor concept using super-heated steam for cooling, now used for containment experiments, situated at Karlstein, Germany
HTGR	High Temperature Gas-Cooled Reactor
HTR	High Temperature Reactor
KONVOI	Siemens-KWU design of EPR
LMFBR	Liquid Metal Fast Breeder Reactor
LWR	Light Water Reactor
NPP	Nuclear Power Plant
PWR	Pressurised Water Reactor
SWR-1000	*Siedenwasserreaktor* (Boiling Water Reactor)-1000
VVER	Russian version of the PWR