Nuclear Safety
NEA/CSNI/R(2016)1
January 2016
www.oecd-nea.org

Computational Fluid Dynamics for Nuclear Reactor Safety-5 (CFD4NRS-5)

Workshop Proceedings
9-11 September 2014
Zurich, Switzerland

BETTER POLICIES FOR BETTER LIVES

NUCLEAR ENERGY AGENCY

NEA/CSNI/R(2016)1
Unclassified

Unclassified **NEA/CSNI/R(2016)1**

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

English text only

NUCLEAR ENERGY AGENCY
COMMITTEE ON THE SAFETY OF NUCLEAR INSTALLATIONS

Computational Fluid Dynamics for Nuclear Reactor Safety-5 (CFD4NRS-5)

Workshop Proceedings
9-11 September 2014
Zurich, Switzerland

English text only

Complete document available on OLIS in its original format

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;

– 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

The NEA Committee on the Safety of Nuclear Installations (CSNI) is an international committee made up of senior scientists and engineers with broad responsibilities for safety technology and research programmes, as well as representatives from regulatory authorities. It was created 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 among NEA member countries. The main tasks of the CSNI 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 reach 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 priority of the CSNI 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 Committee on Nuclear Regulatory Activities (CNRA), which is responsible for issues concerning the regulation, licensing and inspection of nuclear installations with regard to safety. It also co-operates with other NEA Standing Technical Committees, as well as with key international organisations such as the International Atomic Energy Agency (IAEA), on matters of common interest.

TABLE OF CONTENTS

LIST OF ABBREVIATIONS AND ACRONYMS

AIAD	Algebraic interfacial area density
BIT	Bubble-induced turbulence
BNCT	Boron-neutron capture therapy
BPGs	Best practice guidelines
BWR	Boiling-water reactor
CCFL	Counter-current flow limitation
CFD	Computational fluid dynamics
CFX	Computational fluid dynamix (commercial software)
CHF	Critical heat flux
CMFD	Computational multi-fluid dynamics
CNRA	Committee on Nuclear Regulatory Activities
COCOSYS	Containment code system
CSNI	Committee on the Safety of Nuclear Installations
DNB	Departure from nucleate boiling
ECC	Emergency core cooling
ECCS	Emergency core cooling system
FLUENT	Computational fluid dynamics code (commercial software)
FoM	Figure of merit
IAEA	International Atomic Energy Agency
IDDES	Improved delayed detached eddy simulation
LDV	Laser-Doppler velocimetry
LES	Large-eddy simulation
LIF	Laser-induced fluorescence
LIM	Large interface model
LOFT	Loss-of-flow transient
LWR	Light-water reactor
MPS	Moving particle semi-implicit
NEA	Nuclear Energy Agency
NED	Nuclear Engineering and Design (Elsevier journal)
NPP	Nuclear power plant
PCE	Polynomial chaos expansion

LIST OF ABBREVIATIONS AND ACRONYMS *(CONT'D)*

PHWR	Pressurised heavy-water reactor
PIRT	Phenomena identification and ranking table
PIV	Particle image velocimetry
PTS	Pressurised thermal shock
PWR	Pressurised-water reactor
RANS	Reynolds-averaged Navier–Stokes
RPV	Reactor pressure vessel
UQ	Uncertainty quantification
V&V	Verification and validation
VOF	Volume-of-fluid
WAVE	Weighing approach for validation against experiment

EXECUTIVE SUMMARY

Background

This present workshop, the 5th Computational Fluid Dynamics for Nuclear-Reactor Safety (CFD4NRS-5), in the biennial series of such Nuclear Energy Agency (NEA) and International Atomic Energy Agency (IAEA) sponsored events, a tradition which began in Garching in 2006, follows the format and objectives of its predecessors in creating a forum whereby numerical analysts and experimentalists can exchange information in the application of computational fluid dynamics (CFD) to nuclear power plant (NPP) safety and future design issues. The emphasis, as always, was, in a congenial atmosphere, to offer exposure to state-of-the-art (single-phase and multi-phase) CFD applications reflecting topical issues arising in NPP design and safety, but in particular to promote the release of high-resolution experimental data to continue the CFD validation process in this application area.

The reason for the increased use of multi-dimensional CFD methods is that a number of important thermal-hydraulic phenomena occurring in NPPs cannot be adequately predicted using traditional one-dimensional system hydraulics codes with the required accuracy and spatial resolution when strong three-dimensional motions prevail. Established CFD codes already contain empirical models for simulating turbulence, heat transfer, multi-phase interaction and chemical reactions. Nonetheless, such models must be validated against test data before they can be used with confidence.

The necessary validation procedure is performed by comparing model predictions against trustworthy experimental data. However, reliable model assessment requires CFD simulations to be undertaken with full control over numerical errors and input uncertainties. The writing groups originally set up by the NEA have been consistently promoting the use of best practice guidelines (BPGs) in the application of CFD for just this purpose, and BPGs remain a central pillar of the simulation material accepted at this current workshop, as it was at its predecessors. In order to assess the maturity of CFD codes for use in reactor safety and design, it is necessary to establish a database of CFD-grade experimental material; this remains the second pillar of the CFD4NRS series of workshops.

The third pillar is the advancing use of CFD modelling in multi-phase applications, these days known commonly as CMFD. Here, the challenges are considerable. Not only are the governing equations an order of magnitude more complex than for single-phase applications, but a validation database for which there is genuine three-dimensional involvement remains quite sparse. Of course, multi-phase CFD is not the sole province of NPP applications, and important developments are taking place in other industrial arenas, such as in the chemical and processing industries, and in environmental studies. Prudence dictates that the CFD4NRS series of workshops should not provide reporting space for such non-nuclear CMFD applications, but should recognize that links with diverse application areas need to be maintained. After all, there is no capacity for re-inventing the wheel, albeit a multi-phase one.

Scope

A recent IAEA initiative has declared the intention of merging the twin technologies of NPP safety and NPP design within the context of the advancement and application of CFD in nuclear technology. Both of these application areas rely on validation procedures to keep the science trustworthy, and the validation procedures underpinning their reliability have been jointly recognized.

Emphasis in this workshop was placed in the following areas: single-phase and multi-phase CFD simulations with a focus on validation in areas such as: heat transfer, free-surface flows, direct contact condensation, and turbulent mixing. And indeed, many papers were devoted to these issues. The uses of systematic error quantification, and the application of BPGs, were as strongly encouraged as in previous workshops in this series, leading to the rejection of some papers which did not address these issues adequately. Papers submitted related principally to NPP-relevant safety issues, such as pressurized thermal shock, critical heat flux, boron dilution, hydrogen distribution in containments, thermal striping and fatigue, and/or advanced design concepts, such as tight-lattice fuel configurations, and passive safety options. For the first time at these workshops, it was considered that the technology was sufficiently advanced for some discussion of uncertainty quantification (UQ) in CFD to be introduced.

Nonetheless, emphasis should always be placed on the presentation of new experimental data, especially those relating to two-phase flow, for which sophisticated measurement techniques are required, and for which information is scarce. It is also very important to deepen understanding of the physics before starting a numerical analysis. This point was brought out particularly in the context of the OECD-PSI (Paul Scherrer Institut) benchmark exercise based on a test performed at the PANDA facility, which was reported fully at this workshop (see the separate report NEA/CSNI/R(2016)2) .

Experiments providing data suitable for CFD or CMFD validation were also welcomed, though 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 for local measurements. It is now rapidly becoming an obligation for papers describing experiments to include a discussion of measurement uncertainties.

Results and their significance

There were around 110 registered attendees at the CFD4NRS-5 workshop, about 15% down on expectations; the reason for the decline in numbers is not clear. The number of *Extended Abstracts* received for evaluation following the initial announcements was 77. This is similar to previous workshops in the series. All the abstracts were evaluated for suitability by at least two reviewers, and invitations to write a full paper sent out at three hierarchical levels:

- Unconditional (favourable reviews had been received from all the reviewers)
- Conditional (at least one reviewer was unsure of the final acceptance of the paper)
- Guarded (it was anticipated that major revision of the paper would be necessary)

The number of technical papers finally received for evaluation was 56. All technical papers received were evaluated by 2-4 reviewers, each according to journal standards. Of these, 46 were accepted for oral presentation, and 2 to be presented in poster form. The remaining eight papers were withdrawn by their author(s) for various reasons. Five keynote lectures were given, each to introduce the morning/afternoon sessions, as appropriate. In addition, 10 posters were displayed relating to the NEA-PANDA benchmark exercise (for which no accompanying paper was requested).

Factors influencing the slight fall in attendance could be: (i) fewer domestic students; (ii) the subsequent decision of the German, Swiss and Belgium governments to phase out nuclear power following the Fukushima Daiichi nuclear accident in March 2011; (iii) the expense involved in making the trip to Zurich, and the high price of accommodation in the city.

Perhaps reflected in these political issues, though the papers given at the workshop covered a multitude of different nuclear safety topics, the ratio of papers devoted to experimentation to those devoted to CFD analysis was not as well balanced as previously seen in this series, perhaps as a result of withdrawal of central funding.

On the positive side, the acceptance in the use of the best practice guidelines for performing quality CFD computations appears now to be further advanced, if not yet universally accepted, but should be balanced by a programme of experiments capable of producing CFD-grade data for validation purposes. The need for top quality experimental data for improving the models and closure laws, especially for multi-phase flow applications, remains as before. On the experimental side, the pursuit of providing precise details of measurement error (i.e. error bounds on the data) needs to be further enforced.

The next workshop in the series, in 2016, will take place on the campus of the Massachusetts Institute of Technology (MIT), and Professor Emilio Baglietto has already accepted the responsibility for organizing this event.

In the panel session at the close of CFD4NRS-5, delegates confirmed their interest in attending this follow-up workshop, if possible, and considered the two-year interval between workshops to be appropriate. The 2.5 day duration of formal presentations, with a fieldtrip on the afternoon of the third day, was also accepted as an acceptable format.

As is customary at the panel session, which in this case was led by D. Bestion (CEA[1]), summaries were made by the respective session chairpersons of the presentations that were given during the oral sessions, and general comments were invited from the audience. These session summaries are now embodied into the present document.

It was also decided that selected papers from the workshop should constitute a special issue of the Nuclear Engineering and Design (NED) journal, as for previous workshops in this series, and indeed Prof. Y. Hassan, General Editor, confirmed this intention. As a consequence, at the conclusion of the workshop, the individual session chairs were approached to declare which of the papers presented in their Sessions should go forward for archival recognition in this regard. The evaluation process was subsequently set in motion. The expectation is that this special issue will appear in 2015.

Conclusions and recommendations

The Session topics, as expected, were wide and various, including such issues as advanced reactor modelling, flow mixing issues, boiling and condensation modelling, multi-phase and multi-physics problems, plant application, hydrogen transport in containments, advanced measuring techniques, and single and multi-phase flow in reactor cores and sub-channels.

On the organizational side, it was extremely difficult to predict the final number of attendees. This may have been due to several factors, but one could be that the benefits of early registration were not set at a sufficiently attractive level. In previous workshops in the series, around 90% of registrations were received by the “early-bird” deadline. This eased the burden on the organizers to: (i) produce an adequate number of the Book of Abstracts (which needed to be presented to the printers one month before the start of the workshop); (ii) make appropriate arrangements for social events; and (iii) arrange the logistics of the fieldtrip on the last day. This time, the rate of registration remained frustratingly linear up to the deadline date a few days before the opening of the workshop.

There were also a number of very late withdrawals from the official programme. This resulted in organizational chaos. Not only is it unfair to those willing to attend at poster status, which, with appropriate notice, could have been upgraded to an oral presentation, but also means that the Book of Abstracts is already out of date at the time of printing, so neither the session attendees, nor the session chairs, have up-to-date information. In addition, it is not always possible to distribute an erratum in time. It is recommended in the future that a 50% advance, non-refundable, registration fee be charged to each participant awarded an oral presentation. This should effectively put an end to such inconsiderate practices.

1. Commissariat à l’énergie atomique et aux énergies alternatives, France.

General conclusions

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. The message was received that "small is good", and that the workshop should remain rigidly focused on CFD issues, and should not broaden its boundaries beyond this.

- The current format, length and interval between CFD4NRS workshops were generally considered still to be appropriate, as was the rotation of venues worldwide. Hence no changes are proposed in this regard.
- 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 their contemporaries, and, when the opportunity presented itself, visit the test facility on which the exercise was based. This practice will therefore be continued, as far as possible, in the future.
- It was generally considered that the level of quality of the papers had been re-established following the negative comments received at the previous workshop.

Specific recommendations

- The nuclear CFD community should be encouraged to apply and further develop the use of Uncertainty Qualification (UQ) methods in regard to their simulations, including uncertainties arising from the numerical solution procedure, the physical models employed, and in the application of initial and boundary conditions.
- It was noted that CFD is no substitute for properly understanding the basic thermal-hydraulic phenomena involved in the particular numerical analysis being described. The CFD tools should be used instead to quantify the complex interplay between the various physical processes taking place rather than being given the burden of identifying them.
- The analytical presentations at the workshop demonstrated the almost universal application of best practice guidelines (BPGs) in producing CFD simulations presented these days, including the use of higher-order differencing methods, where appropriate, or possible.
- In reactor applications, the need for grid sensitivity studies still has to be balanced against the availability of adequate computational resources. This point was brought out specifically in the context of the PANDA benchmark exercise.
- The need for error bars on experimental data has to be reinforced. Any subsequent analysis of the numerical data, vis-à-vis the test measurement, is rendered almost worthless for validation purposes without this pre-requisite.
- It is recommended that future test data offered to this series of workshops must include estimates of measurement error. It appears that best practice guidelines on the presentation of experimental data does not yet exist. In the context of further workshops in this series, a strong message of correction must be sent in this regard.

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.

Keynote lectures

1. M. Andreani; Paul Scherrer Institut (PSI), Switzerland
   Synthesis of results of the OECD-PSI blind benchmark exercise on erosion of a stratified layer by a buoyant jet in a large volume.
2. C. Boyd; US NRC, USA
   Perspectives on CFD Analysis in Nuclear Reactor Regulation.
3. Y. Hassan; Texas A&M, USA
   Multi-scale, full-field measurements and near-wall modeling of turbulent subcooled boiling flow using innovative experimental techniques.
4. D. Lucas; HZDR, Germany
   A strategy for the qualification of multi-fluid approaches for nuclear reactor safety.
5. M. Z. Podowski; RPI, USA
   Model Verification and Validation issues for multiphase flow and heat transfer simulation in reactor systems.

Poster papers

1. S. Mimouni , C. Baudry, M. Guingo, M. Hassanaly, J. Laviéville, N. Méchitoua, N. Mérigoux, A. Schumm; EDF, France
   Modelling of Mitigation Means in a Multi-Compartment Geometry with a CMFD Code
2. A. Gineau, E. Longatte, D. Lucor, P. Sagaut ; CEA and Sorbonne University, France
   Macroscopic Modeling of Fluid-Structure Interaction in Cylinder Arrays using the Theory of Mixtures

Poster papers on the OECD-PSI Benchmark

C. Boyd, J. Thompson	US NRC, USA
S. Yu. Grigoryev, A. S. Filippov	IBRAE, Russia
G. Jiménez	UPM, Spain
T. Nishimura	NSR, Japan
R. Mukin	PSI, Switzerland
S. Abe	JAEA, Japan
A. Skibin, L. Golibrodo, A. Nikolaeva, A. Schukin, A. Krutikov, A. Nechaev, Y. Nadinskiy, V. Volkov	OKB "GIDROPRESS", Russia
R. Huhtanen	VTT, Finland
S. A. Karabasov, A. M. Zaitsev, M.A. Zaitsev	IBRAE, Russia

TECHNICAL PROGRAMME (PRESENTED ORALLY)

Session 1: CONTAINMENT 1

Co-Chairs: M. Andreani (PSI, Switzerland), S. Mimouni (EDF, France)

1. R. Kapulla, G. Mignot, S. Paranjape, D. Paladino
   Stably Stratified Helium Layer Erosion by a Vertical Buoyant Helium-Air Jet
2. S. Kelm, M. Ritterath, H.-M. Prasser, H.-J. Allelein
   Application of the Mini-PANDA Test Case of Erosion of a Stratified Layer by a Vertical Jet for CFD Validation
3. A. Belt, E.A. Reinecke, St. Kelm, A. Hundhausen, H.-J. Allelein
   Design, Construction and Pretest Results of the CFD-Grade Condensation Test Facility SETCOM
4. A. Filippov, S. Grigoryev, N. Drobyshevsky, A. Kiselev, A. Shyukin, T. Iudina
   CMFD Simulation of the ERCOSAM PANDA Spray Tests PE1 and PE2

Session 2: CORE STUDIES

Co-Chairs: H.-M. Prasser (ETHZ, Switzerland), U. Bieder (CEA, France)

1. A. Saxena, R. Zboray, H.-M. Prasser
   Simulations and Measurements of Adiabatic Annular Flows in a Triangular Tight Lattice Nuclear Fuel Bundle Model
2. M.H.A. Piro, B.W. Leitch, F. Wassermann, S. Grundmann, C. Tropea
   Progress in On-Going Fluid Dynamic Investigations of a Deformed CANDU Fuel Bundle
3. K. Takase
   Numerical Simulation of Turbulent Heat Transfer behind a Spacer with Small Ribs in a Sub-Channel
4. A. Saxena, H.-M. Prasser
   Liquid Film Modeling in BWR Subchannel with Spacers

Session 3: V&V 1

Co-Chairs: C. Boyd (US NRC, USA), B. L. Smith (PSI, Switzerland)

1. B. Gaudron, H. Cordier, S. Bellet, D. Monfort
   Using the PIRT to Represent Application and Validation Domain for CFD Studies
2. M. Tanaka, S. Ohno, H. Ohshima
   Development of V2UP (V&V plus Uncertainty Quantification and Prediction) Procedure for High Cycle Thermal Fatigue in Fast Reactor – Framework for V&V and Numerical Prediction
3. D. Bestion
   Needs of Validation Data for CFD Simulation of Two-Phase Flow in a LWR Core

Session 4: CONTAINMENT 2

Co-Chairs: G. Zigh (US NRC, USA), S. Kelm (FZJ, Germany)

1. L. Ishay, G. Ziskind, A. Rashkovan, U. Bieder, J. Brinster
   Simulation of the LOWMA-3 MISTRA Experiment
2. O. Iliev, K. Steiner, A. Zemitis, W. Klein-Hessling, M. Sonnenkalb, M. Freitag
   Towards a combined CFD/LP code approach for containment simulations
3. M. Freitag, E. Schmidt, S. Gupta, G. Poss
   Simulation benchmark based on THAI experiment on dissolution of a steam stratification by natural convection

Session 5: TWO-PHASE FLOW 1

Co-Chairs: M. Z. Podowski (RPI, USA), S. Lo (CD-ADAPCO, UK)

1. M. Tekavčič, B. Končar, I. Kljenak
   Simulation of Flooding Waves in Vertical Churn Flow
2. E. Krepper, R. Rzehak, D. Lucas
   Extended Validation of a Baseline Closure Model
3. Dehbi, F. Han, J. Kalilainen
   Large-Eddy Simulations of Flow inside a Cubical Differentially-Heated Cavity under Realistic Boundary Conditions

Session 6: ADVANCED REACTORS

Co-Chairs: E. Laurien (UniStuttgart, Germany), Y. Hassan (TAMU, USA)

1. M. Böttcher
   CFD Simulation of Heat Transfer in Liquid Metal Flow by using an Innovative Re Analogy Approach
2. G. Matulik, S. Tóth, A. Aszódi
   CFD Analysis of the L-STAR Facility and Validation of the ANSYS CFX Code for Gas-Cooled Reactor Simulations
3. J. Chicheportiche, C. Fournier
   Numerical Evaluation of Gas Entrainment Onset Conditions for Sodium-Cooled Fast Reactor: Application to BANGA Experiment

Session 7: TWO-PHASE FLOW 1

Co-Chairs: D. Bestion (CEA, France), D. Lucas (HZDR, Germany)

1. P. Coste, N. Mérigoux
   Two-Phase CFD Validation: TOPFLOW-PTS Steady-State Steam-Water Tests 3-16, 3-17, 3-18, 3-19
2. M. Sharabi, B. Niceno, V.F. González-Albuixech, M. Niffenegger
   Computational Fluid Dynamics Study of Pressurized Thermal Shock Phenomena in the Reactor Pressure Vessel

3. N. Mérigoux, J. Laviéville, S. Mimouni, M. Guingo, C. Baudry
   Reynolds Stress Turbulence Model applied to Two-Phase Pressurized Thermal Shocks in a Nuclear Power Plant

Session 8: EXPTL. TECHNIQUES

Co-Chairs: H.-M. Prasser (ETHZ, Switzerland), Y. Hassan (TAMU, USA)

1. F. Wassermann, S. Grundmann, C. Tropea, M.H.A. Piro, B.W. Leitch
   Magnetic Resonance Velocimetry Measurements of the Flow through a Fuel Rod Bundle
2. G. Lundqvist, K. Angele, P. Veber, L. Andersson
   Validation of CFD with FSI: Damping of Forced Rod Vibrations
3. J. Xiong, Y. Yu, X. Cheng
   Three-Dimensional LDV Measurement of Turbulent Flow in a 6x6 Rod Bundle

Session 9: PLANT APPLICATION

Co-Chairs: G. Yadigaroglu (ASCOMP, Switzerland), S. Bellet (EDF, France)

1. T. Toppila, J. Kuopanportti, T. Rämä
   Assessment of CFD Model for LOVIISA NPP Coolant Mixing Studies using Data of Steam Safety Valve Tests
2. V. Petrov, B. Kendrick, D. Walter, A. Manera, J. Secker
   Prediction of Crud Deposition on PWR Fuel using a State-of-the-Art CFD-Based Multi-Physics Computational Tool
3. J. Yan, P. Yuan, L.D. Smith, Z. E. Karoutas, P. Joffre
   PWR Fuel Sub-Channel Thermal Mixing CFD Model Development and Validation

Session 10: BOILING FLOW

Co-Chairs: B. Gaudron (EDF, France), P. Coste (CEA, France)

1. Y. Liao, D. Lucas, E. Krepper, R. Rzehak
   Assessment of CFD Predictive Capacity for Flash Boiling
2. S. Mimouni, C. Baudry, M. Guingo, J. Laviéville, N. Mérigoux, N. Méchitoua
   Computational Multi-Fluid Dynamics Predictions of Critical Heat Flux in Boiling Flow
3. S. Vahaji, S. C. P. Cheung, J. Y. Tu, G. H. Yeoh
   Modeling Sub-Cooled Boiling Flow at Elevated Pressures — Evaluation of Mechanistic Approach

Session 11: V&V 2

Co-Chairs: D. Lucas (HZDR, Germany), Y. Yassin (TAMU, USA)

1. T. Höhne
   Validation of Closure Models for Interfacial Drag and Turbulence of Horizontal Segregated Flows
2. M. Ishigaki, S. Abe, Y. Shibamoto, T. Yonomoto
   Numerical Simulation of Thermal Flow with Steam Condensation on Wall using the OpenFOAM Code
3. S.-K. Park, J.-S. Lim, Y.-H. Jung
   CFD Analysis Methodology for Sub-Channel of Fuel Rod Bundle

Session 12: UQ

Co-Chairs: M. Tanaka (JAEA, Japan), B. Niceno (PSI, Switzerland)

1. Barthet, O. Brero, H. Cordier, R. Camy
   Uncertainty Quantification for a Pressurized Thermal Shock Experiment using the Wave Method
2. Badillo, B. Ničeno, J. Fokken, R. Kapulla, J. Ko, J. Galpin
   Uncertainty Quantification of the Effect of Random Inputs on Computational Fluid Dynamics Simulations of the GEMIX Experiment using Metamodels
3. Papukchiev, G. Lerchl, C. Geffray, R.-J. Macián, M. Jeltsov, K. Kööp, P. Kudinov
   Coupled 1D-3D Thermal-Hydraulic Simulations of a Liquid Metal Experiment Supplemented by Uncertainty and Sensitivity Analysis

Session 13: PRIMARY CIRCUIT

Co-Chairs: U. Bieder (CEA, France), B. L. Smith (PSI, Switzerland)

1. G. H. Lee, Y. S. Bang, A. J. Cheong
   Comparative Study for Modeling Reactor Internal Geometry in CFD Simulation of PWR and PHWR Internal Flow
2. P. Prusiński, L. Kozioł, T. Kwiatkowski, S. Potempski, A. Prusiński
   LOFT in BNCT Converter
3. M. Kuschewski, R. Kulenovic, E. Laurien
   Experimental Investigation of Stratified Pipe Flow Mixing in a Horizontal T-Junction

Session 14: TWO-PHASE FLOW 2

Co-Chairs: B. Niceno (PSI, Switzerland), S. Lo (CD-ADAPCO, UK)

1. S. Mimouni , C. Baudry, R. Denèfle, S. Fleau, M. Guingo, M. Hassanaly, J. Laviéville, N. Méchitoua, N. Mérigoux
   Multi-Field Approach and Interface Locating Method for Two-Phase Flows in a Nuclear Power Plant
2. J. Lehmkuhl, S. Kelm, M. Bucci, H.-J. Allelein
   Improvement of Wall Condensation Modeling with "Suction" Wall Functions for Containment Application
3. T. Ma, T. Ziegenhein, D. Lucas, E. Krepper, J. Fröhlich
   Large-Eddy Simulations for Dispersed Bubbly Flows

Session 15: BDBA STUDIES

Co-Chairs: S. Mimouni (EDF, France), G. Yadigaroglu (ASCOMP, Switzerland)

1. S. Park, B.-I. Jang, H. J. Kim
   3-D Simulation of Plunging Jet Penetration into a Denser Liquid Pool by the RD-MPS Method
2. R. Mukin, A. Dehbi
   Simulation of Flow on the Secondary Side of a Steam Generator Bundle

Session 16: OECD–PSI BENCHMARK WRAP-UP

Co-Chairs: B. L. Smith (PSI, Switzerland), M. Andreani (PSI, Switzerland)

1. B. L. Smith
   Overall Organization of the Benchmark Exercise
2. M. Andreani
   Brief Summary of the Conclusions of Benchmark Synthesis
3. OPEN DISCUSSION

TECHNICAL SESSION SUMMARIES FROM THE CO-CHAIRS

Session 1: CONTAINMENT 1

The Session included four papers, three of them focused on gas stratification and mixing, and one presenting a new experimental facility for investigating film condensation.

The first paper (Mignot et al.) illustrates in detail the experiment in the PANDA facility that was used for the NEA-PSI CFD benchmark exercise. In this test the gradual erosion of a helium-rich layer at the top of a vessel containing pure air beneath by means of a vertical air-helium jet was studied. Special attention was paid to an accurate characterization of initial and boundary conditions. In particular, off-line PIV measurements were carried out to provide detailed information on exit profiles of mean velocity and RMS of the velocity. Helium concentrations and gas temperatures time-histories at various locations provided a complete picture of the stratification erosion, and the PIV measurements at three positions offer a valuable insight in the interaction of the of the jet with the density interface. This wealth of information, only partly used for the benchmark exercise, makes the test very valuable for follow-up CFD validation studies.

The second paper (Kelm et al.) also addresses a gas mixing process in an initially stratified fluid domain. It presents the results of the simulations of two tests in the MINI-PANDA facility using the ANSYS CFD code in the context of a more comprehensive investigation on the capability of the code to simulate stratification erosion for a variety of conditions. In particular, the interaction Froude number is used for classifying the flow regimes (inertia-controlled, buoyancy controlled, and transition regime) and showing the relevance of the tests in MINI-PANDA for the code validation. The two tests, especially the one with a lower Froude number, could be successfully simulated. A key aspect of the validation was a systematic use of the Best Practice Guidelines, which contributed to the reduction of numerical and modelling errors at the price of only a small increase of computational costs. The only uncertainty remaining was the neglect of some internals (instrumentation supporting rods). An important consideration resulting from the analysis was related to the complementarity of the experiments used for the validation. In fact, densely instrumented, small facilities (like MINI-PANDA) are very suitable for code validation, but the low values of the Reynolds number that can be achieved result in an overemphasised role of molecular diffusion. On the other hand, at large scale, turbulence modelling becomes more crucial. For the overall validation of the codes it is therefore necessary to use also data from tests in larger facilities.

The third paper (Belt et al.) presented the design and the results of the first commissioning tests of a new large facility (SETCOM – Separate Effect Test for COndensation Modelling) devoted to the study of film condensation. In particular, the objective of the new experiments is to provide information on the wall boundary layer, which could be used for formulating a more adequate law of the wall for CFD models of film condensation. The facility is therefore equipped with advanced, non-intrusive instrumentation, aiming to measure velocities and other variables affecting heat/mass transfer processes close to the condensing wall. The facility can be rotated, so that vertical as well as horizontal and inclined configurations can be investigated. The facility is designed to permit the study of the entire range of heat/mass transfer regimes of interest for containment analysis, thus including forced convection, natural convection and mixed convection. Lastly, the first commissioning experiments confirmed that the thermal response of the facility matched the technical specifications.

The fourth paper (Filippov et al.) addressed the simulation of two tests in the PANDA facility using the FLUENT code. The two tests included various phases, the first period featuring injection of steam and helium, which resulted in stable stratification. The last phase of the transient investigated the effect of water spray injection on pressure evolution and gas mixing. The two tests have been started with different initial conditions, and different spray nozzles were used in the two experiments. The model for the spray considered in detail the available information on droplet size and velocity distributions, which depended on the nozzle type. The stratification build-up during the phase before spray operation could be reproduced quite well, the only important parameter showing a notable discrepancy with the data being the peak helium concentration, which was overpredicted. Depressurisation and gas mixing caused by the spray could also be predicted reasonably well, and the different patterns of stratification breakup in the two tests could be captured. Sensitivity studies revealed that the most important parameters affecting gas mixing were the spray angle and the initial droplet velocity. It was argued, however, that these two parameters characterizing the boundary conditions at the spray nozzle would play a smaller role in the analyses of a NPP, because the dome of a real containment is much higher than the vessel of the experimental facilities, and thus the larger droplet fall distance would reduce the effects of the details of the droplet source.

Session 2: CORE STUDIES

This Session consisted of four papers. The first speaker presented a simulation of the annular flow in a system of four neighbouring sub-channels of a fuel rod bundle with triangular tight-lattice geometry, equipped with a spacer grid with three vanes. Large-eddy Simulations (LES) coupled to a volume-of-fluid (VOF) approach to predict the dynamic gas liquid surface in a high void fraction case was used to simulate the turbulent wavy film flow along the surface of the fuel rods. This is a very time consuming technique. The commercial code STAR-CCM+ was applied. The results of the calculations compare well to measurements of the time-averaged liquid film thickness and its modulation by the functional spacer, which were carried out by tomography with cold neutrons at the ICON beam-line of the SINQ spallation neutron source of PSI.

The second paper provided an interim status report of a project aiming at the measurement of the flow field in a CANDU bundle deformed after dryout. The bundle starts to settle by creep, which causes a gradual departure from the nominal geometry. In the result, the free cross-section at the bottom of the channel is reduced, while a bypass gap opens at the top. The authors pursue the measurement of the three-dimensional flow field in such a bundle by means of a Nuclear Magnetic Resonance imaging technique, namely by Magnetic Resonance Velocimetry. In the talk, the results of measurements at an undeformed reference bundle were presented and used for CFD code validation. Very impressive results were obtained. The bundle was manufactured from non-ferromagnetic material by 3D printing; a medical MRI machine in a hospital was used. The final purpose is the investigation of the coolability of the deformed bundle. In the presentation, the method of obtaining a realistically deformed bundle by heating a specimen in a furnace was presented, as well. This was a very good paper.

In the third paper, simulations of the supercritical water flow in a heated sub-cannel of a tight-lattice arrangement of fuel rods downstream of a spacer grid with wing-shaped vanes were reported by the author. Japan Atomic Energy Agency has developed an analysis code to obtain precise thermo-fluid properties of supercritical fluids. The heat transfer calculations were performed using a newly developed CFD code. The author was able to predict a heat transfer enhancement by the proposed spacer, which was characterized by a decreasing maximum cladding surface temperature.

In the final paper in the Session, work was described dealing with coupling of a simplified 3D RANS simulation of the flow in the gas core flow with a genuine 2D liquid film flow model to predict the annular flow in a pair of neighbouring sub-channels of a BWR bundle with a spacer grid with mixing vanes. The simplification of the RANS approach, carried out with STAR-CCM+ (steady-state k-omega turbulence model) lies in neglecting the water film. The sub-channel geometry and the spacer grid were fully represented, inducing adequate mixing. An in-house film flow model was coupled to the CFD calculation via the 2D shear stress and pressure fields sampled at the wall in the output data of the RANS calculations. The results were compared to experiments at an in-house test loop. They show encouraging agreement with the measured film thickness profiles. The present film model has overcome some of the limitations of an earlier film flow model, which neglected the inertia of the water in the film. The new model is based on a 2D formulation of the Navier-Stokes equations.

In summary, the papers presented in the Session were of high quality and very interesting. A special highlight was the presentation of the MRV studies of the velocity field in a CANDU bundle (Paper 2.4). This paper deserves to be included into a special NED issue. It should be combined with the paper on the measuring method from Session 8 (Paper 8.2), whereas the emphasis should be put on the measuring results. Again, concerning Paper 8.2, general explanations of MRV and MRI should be included in a concise way.

Session 3: V&V 1

There were three papers presented in this Session. All dealt with strategic options for performing quality verification and validation (V&V), coupled with an uncertainty quantification (UQ) analysis. The first paper described an innovative approach developed at EDF, France for identifying the regions of overlap between validation and application domains derived from a PIRT. A figure of merit (FoM) was defined, the physical phenomena, main parameters and dimensionless numbers, all displayed graphically to illustrate their interaction. The authors showed how their approach could be used for a boron dilution problem. The paper gave rise to some lively discussions on the benefits and limitations of the approach.

The second paper focused on V&V and UQ in CFD, which had been developed by the JAEA in Japan in the context of thermal striping issues associated with sodium-cooled reactors. An interesting observation was the fact that the scaling methodology for V&V originally laid down by Zuber and co-workers in 1998 and the much more up-to-date ASME V&V 10 (2012) were structurally very similar. A flow chart of the process and some limited results were presented. A PIRT and FoM featured strongly in the approach.

The third paper, given by D. Bestion, CEA France, expounded the extra challenges associated with V&V/UQ strategies applied to two-phase flow applications. He made the point in particular that the 3D capability of CFD requires V&V also in 3D. However, here there is a definite lack of knowledge. For example, most boiling models are based on empirical correlations derived from 1D pipe flow. Though suitable strategies for V&V/UQ can be laid down, as with 3D single-phase flows, it is clear that the present experimental database is insufficient. Proposals and the rationale for new experiments were addressed.

Together, the three presentations indicated the emergence of a common strategy for qualifying CFD simulations in nuclear technology, simulations that can then be used in the regulatory environment. There needs to be considerable advances made in computer performance and/or algorithm efficiency for UQ to be a tenable approach in single-phase reactor applications, and considerable extra effort in 3D experimentation, with local measurement possibilities, to justify CFD in multi-phase situations.

Session 4: CONTAINMENT 2

The session comprised of three presentations, two on gas mixing in typical containment flows and one on a coupled CFD-LP approach in which a dedicated CFD approach was developed to predict the behaviour of water pools and complement a LP containment model of the gas phase.

The first paper by Ishay et al. addressed the systematic preparation and first validation of a CFD approach based on the LOWMA3 experiment, which is part of the so-called ”PANDA – MISTRA common test” performed during the NEA SETH-2 project. The test scenario is similar to the current PANDA benchmark exercise on the erosion of a stratified layer by a vertical jet. The authors presented a comprehensive study on different modelling aspects, in particular the effect of spurious velocities on the molecular diffusion process, turbulence modelling on the jet flow and mixing progress and various numerical parameters before they applied the model to the experimental transient.

The second presentation titled by Zemitis et al. introduced the dedicated 3D CFD model CoPool (Containment Pool). The model is based on 3D Navier-Stokes equations with Boussinesq approach for the liquid phase and 3D heat conduction equations for the immersed structures. The code is intended to be used to complement the LP code COCOSYS for detailed 3D simulations of processes in large containment water pools, like the wetwell of a BWR. Following a discussion of the main numerical and modelling features of CoPool, the coupling scheme based on PVM was presented. A first validation, based on the GES Water Pool Experiments and first promising application results showed that a thermal stratification of a water pool can be resolved by the coupled approach.

The paper by Freitag et al. summarized a simulation of a benchmark experiment carried out within the German CFD alliance. The presenter summarized the experimental campaign performed at the THAI facility, in which a natural circulation flow was established by differential heating of the THAI vessel wall sections. A stratification is built up by means of rapid injection of steam, which then slowly mixes by the recovering buoyant flows. The different CFD and LP modelling approaches adopted by the benchmark participants were introduced and the blind and open simulation results to the experiment compared. There was visible progress made from the blind to the open phase of the benchmark, which was related mainly to the treatment of the thermal boundary conditions of the heated and cooled wall section.

Session 5: TWO-PHASE FLOW 1

There were three papers in this session. Only a summary of Paper 2 by Krepper et al. has been received.

This paper describes the initiative of the Rossendorf team in defining the baseline closure model and their work in validating this closure model against the CFD-grade experimental data from the MT_Loop. The closure models in defining the force and turbulence for bubbly flows were described. Measure data from the MT_Loop were used in the validation exercise. The comparison results show the Baseline Closure Model was not able to match some of the measured data, and in particular the model failed to capture the effects of pipe inclination. The role of the lift force was investigated, and it was found that a reduction of the lift force may be required to obtain better correspondences.

Session 6: ADVANCED REACTORS

The session was announced with contributions on liquid metal flow, gas-cooled reactors, and sodium-cooled fast reactors. The first speaker (Böttcher) could not give his presentation due to technical reasons (data file of his presentation not available). The second speaker (Matulik) did not appear, therefore this presentation was also not given. The third speaker (Chicheportiche) gave a very interesting presentation about experiments in the BANGA facility of CEA Grenoble, and his analysis and numerical simulation using TRIO_U. Flow with a free surface along a channel is combined with a downward suction flow. Due to the downdraft, a strong deformation of the free surface occurs, combined with a vertical circulation. The flow can be simulated using the large-eddy method. Further analytical analysis leads to parametric criteria about the onset of the deformation. These criteria use idealized conditions and geometries. They are validated using the experiments. To take account of more complex geometries, and the superposition of the flow, more CFD simulations are necessary. An extended discussion demonstrated the great interest of the audience for this flow problem.

Session 7: PTS

Three papers were presented in this session.

The first was an application of single-phase CFD to two different single-phase reactor PTS scenarios with equal or non-equal ECCS injection into two circulation loops. The ANSYS-FLUENT code was used for the simulations. Boundary conditions were provided by a parallel RELAP-5 system code calculation. The effect of the ECCS injection on the thermal and mechanical loads on the wall of the RPV was found to be more severe in the case of non-symmetrical injection. The predictions of the thermal and mechanical loads with the 'System+CFD' approach are significantly different than those obtained with the system code alone, showing the benefits of a better space resolution. The CFD results showed high three-dimensional flow behaviour in the downcomer, which confirms the inadequacy of one-dimensional code treatments to properly predict the thermal loads generated during ECC injection scenarios. This work shows the maturity of the CFD application. However, to give full confidence in the result, this work should be complemented by information on the assessment and on the application of BPGs.

The other two papers in the session were devoted to PTS in two-phase scenarios. They focused on the validation of the models implemented in the NEPTUNE-CFD code. These models have been developed and validated for at least ten years against many separate-effect tests, and against combined-effect tests like the recent TOPFLOW-PTS tests. It was found that the most important phenomena are well represented qualitatively and quantitatively. For example, the TOPFLOW tests showed that the flow in the inclined injection pipe could be either pure liquid or a stably-stratified steam/water mixture, or even unstably-stratified, depending on the flow rate. The actual flow regime has a strong influence on the mixing process. It was shown that, provided sufficient care is taken to the nodalization of the nozzle region, these different flow regimes can be captured by the CFD code. It was also shown that a Rij-ε model performs better than a k-ε model, and that mesh convergence could be obtained with a reasonable number of meshes: about one million meshes is sufficient. If we compare the present results to those presented in the previous CFD4NRS workshop from 2006, one can say that the two-phase PTS simulation issue is not solved and closed, but the application of two-phase CFD to PTS is now very close to the realization. There was strong scepticism at the start of this activity, but the authors of these two papers did not follow this scepticism, and have been proved right. For both papers, a rigorous strategy was followed from the beginning with a PIRT analysis, collection of the relevant experimental data, the benchmarking of several modelling approaches for the free surface (one-fluid + ITM or two-fluid + LIM) and for turbulence modelling (k-ε, Rij-ε, LES, VLES), and an extensive validation with attention paid to mesh convergence.

This confirms the interest of continuing our common efforts to promote the use of CFD for nuclear reactor safety issues.

Session 8: EXPERIMENTAL TECHNIQUES

Three papers were presented in this session. The first presenter explained the method of Magnetic Resonance Velocimetry, which had been successfully applied to map the flow field in a CANDU bundle. A medical MRI machine was used here as part of the instrumentation, the bundle mock-up being fabricated on a 3D printer. The audience was introduced into this novel measuring technique. It was a very nice, motivating presentation, raising the appetite for more applications of MRI techniques in the future in this area. The paper complemented Paper 3 in Section 2.

The second paper was an experimental study of the damping of externally induced oscillations of a coaxial metallic rod inside a vertical pipe. The pipe is sometimes kept empty and at other times filled with water. In the latter case, experiments were performed with stagnant water, and later with flow at two different flow rates. The test channel operates at ambient conditions. The rod is set into motion by pulling it to one side in the centre and then releasing it. Damping was seen to be nearly two orders of magnitude higher in the presence of water compared to an empty test pipe. Measured frequencies show the effectiveness of the added mass, which in fact decreases with increasing water velocity. A good agreement with FSI simulations using ANSYS-FLUENT was found. The authors recommend that their data are useful for code validation work, and declare that they can be obtained free of charge upon application. In the outlook, they announced that the test section will be modified for future experiments, in which a circular rod will be positioned between four fuel box corners to study the fluid elastic instability due to the small leakage flow between the boxes.

In the final paper, the authors present their very carefully accomplished LDV measurements of all three velocity components in single-phase flow conditions in a rod bundle with spacer grids. A laser was used to 'shoot' into the gaps between the rods and, in a second measuring series, from the outlet side in the axial direction into the bundle. Measurements were made of the time-averaged velocity field, the turbulent kinetic energy, and all accessible components of the Reynolds stress tensor at the accessible positions. The data are very valuable for CFD code validation.

In summary, all three papers were very interesting, and of high quality. For a special issue of NED, the third paper particularly was outstanding. The results shown are of high value for CFD code validation purposes.

Session 9: PLANT APPLICATION

The papers in this session covered one area in which CFD has been most often used: namely, boron mixing in the primary system, and two fuel-bundle related problems — crud deposition and sub-channel mixing.

The first paper, presented by T. Topilla from Fortum in Finland, focused on the practical difficulties encountered when boron mixing is computed for a real reactor system rather than for an experimental facility. The main difficulties include the necessity to take into consideration the *as built* plant configuration (where small deviations, such as the angle of the cold-leg penetration into the reactor pressure vessel could make a significant difference), the need to conduct all the work under quality assurance procedures, and to embody CFD best practice guidelines, and the employment of grid refinement studies, etc. A commercial code was used with 7 to 30 million cells in this study.

The second paper, presented by A. L. Manera of the University of Michigan, dealt with the prediction of crud deposition on fuel rods in an operating reactor, which is a complex multi-physics phenomenon. Within the context of the US CASL project, considerable effort has been expended to couple CFD computations with both neutron and chemistry codes in pursuit of this issue. Mesh convergence studies, and a comparison of CFD results against plant data (in term of measurements of crud thickness), indicate that CFD is a useful tool in improving the safety for reactors currently in operation by understanding the complex phenomenon of crud deposition. First results from the associated simulations are encouraging, and in good agreement with real life operations, all leading to a better understanding of the phenomenon.

The third presentation was given by L. D. Smith from Westinghouse, and addressed fuel-bundle sub-channel mixing; CFD being used to improve system code simulation capability. In particular, it was demonstrated how the vendor can make use of CFD to increase the safety level of new fuel designs. The study was respected the application of BPGs in the study, notably in regard to mesh convergence, and in validating the CFD results based on different grid configurations against full-scale data based on data from a 5x5 fuel bundle test.

Session 10: BOILING FLOW

This session consisted of three presentations. The first was by Yixiang Liao from HZDR, Germany. The paper dealt with the assessment of predictive capacity of CFD for flash boiling. The physical phenomena associated with flashing were first identified. A literature survey was undertaken concerning the appropriateness of the available two-fluid models to describe the phenomenon, mostly in their 4-equation and 5-equation formulations. The main issues concerning the modelling were identified, these being, slip and non-drag forces, the heat transfer coefficient, and nucleation. The different models were compared, and simulations tested against two experiments: BNL for steady-state critical flow, and TOPFLOW for the pressure transients.

The second presentation was given by Stephane Mimouni from EDF. This dealt with one historical issue of nuclear thermal hydraulics: namely, the prediction of the Critical Heat Flux (CHF). Calculations were carried out using NEPTUNE_CFD in simple geometries. Crucial to this work is that a fourth flux is added to the three classical fluxes of the Kurul and Podowski model when the DNB criterion is encountered. After first demonstrating mesh independency, many experiments were subsequently simulated, investigating the effects of mass flux, steam quality, diameter, pressure and fluid properties on CHF. The mean relative error of these mechanistic CHF predictions was reported to be 8.3%.

The third presentation was given by Professor Gwan Yeoh from the University of New South Wales, Australia. The paper dealt with sub-cooled boiling flow modelling. In the model presented, the active nucleation site density is evaluated from a fractal analogy approach. The bubble lift-off, sliding diameters and the departure frequency are all evaluated by means of a force balance method. Interest of this new modelling approach is demonstrated by comparisons with three experiments conducted at pressures of 1, 2 and 5 bar.

In summary, the three papers presented in this session use established models, and compare predictions with various experiments, under various conditions, which is good validation practice. New models are also developed, and there is a valuable literature survey reported. The importance of the effort to develop mechanistic models rather than rely on purely empirical models has to be underlined. Considering the number of phenomena involved in boiling flows, one may suggest using the PIRT method to get a synthetic view of the state of the art, and then to focus on the relevant issues subsequently identified.

Session 11: V&V 2

The three papers of this session addressed validation issues for models on segregated flows, steam condensation on walls, and sub-channels flows in fuel rod bundles.

The first contribution, given by Th. Höhne from HZDR, focused on the algebraic interfacial area density (AIAD) model, which has been implemented into the ANSYS-CFX code. The model allows different flow morphologies to be considered, including the occurrence of bubble and drops adjacent to large gas liquid interfaces. In addition, an improved drag model, as well as models for turbulence damping and sub-grid wave turbulence, is also available. The options for closure models, including all tuneable constants, are fixed as a result of a baseline model strategy. Examples for validation were presented. These included comparisons of simulation results with experimental data from the Fabre experiments on adiabatic flow in a horizontal channel, the HAWAC channel of HZDR and the CCFL data from the HZDR hot-leg model. The AIAD model and the large interface model (LIM) seem to be the most advanced models presently available for the simulation of segregated flows in the frame of the two-fluid approach. The baseline model concept is important to obtain models with predictive capabilities in future.

The second presentation was given by M. Ishigaki from the Japan Atomic Energy Agency. He described a new model, and its validation, for steam condensation on cold walls. Background to this work is the multi-phase and multi-component flow in a containment under severe accident conditions. The model, which has been implemented into the open-source code OpenFOAM, considers the diffusive mass fluxes of steam and non-condensable gases towards the wall; special attention is paid to the velocity and temperature distributions close to the wall. In this work, the liquid film at the wall is not explicitly modelled. Two test cases were used for the validation of the model: the data of Park et al., 1996 (downward flow of a steam-nitrogen mixture along a vertical channel with a cold wall), and the data of Kang and Kim, 1999 (downward flow of a steam-nitrogen mixture along an inclined channel with a cold wall at the lower side).

In the last contribution of this session, S.-K. Park from KEPCO Nuclear Fuel presented CFD simulations of single-phase flow in fuel rod bundles. To set up the CFD model, a 4 x 4 rod bundle configuration was considered. Simulations were carried out using the FLUENT code. A 1/8th sector model of the bundle was considered, making use of symmetry assumptions regarding the bundle geometry. For the turbulence modelling, both the k-ε and the k-ω models were applied. The setup was then applied to a thermal mixing test featuring a 5 x 5 bundle with non-uniform radial heating. Results were compared with experiments and simulation results obtained also using a sub-channel code. Overall, results showed similar trends. However, CFD validation of these test cases may be limited by the available experimental data, which are not detailed enough.

For all three contributions to this session, specific model setups were presented, and simulations results compared with experimental data. Some measure of agreement was obtained, but in some cases there are clear deviations between simulation results and experimental data, demonstrating the need to develop more substantiated model setups in future. In any case, it is important to perform the model validation against data from more than one experiment, but without modifying the model setup for the single test cases.

Session 12: UQ

In the first paper, an uncertainty quantification (UQ) method called the weighing approach for validation against experiment (WAVE) was proposed. This method is linked tightly to the PIRT in verification and validation. The WAVE method could take into account the bias error of CFD results at reactor scale, the external input uncertainties, and the numerical uncertainties due to the numerical model. Although WAVE might be a useful tool to estimate UQ at plant scale, more discussions would be required in application of the scalability hypothesis from sub-scale mock-up integral tests to the reactor scale.

In the second paper, the polynomial chaos expansion (PCE) scheme was used to quantify uncertainty. The PCE scheme considers the propagation of uncertainties in the system. At the present time, unfortunately, it has only been applied to the simple problem of two-layer thermal mixing. Although it might be very challenging, this method is strongly expected to solve the practical problem as a next step in the study.

In the last paper, a coupling methodology between a commercial CFD code and an in-house system code was applied to a simple experimental configuration of natural circulation phenomena. The test loop consists of a circulation loop, a heater in the lower position of the loop, a cooler at a higher position of the loop, and a vertical bypass line with a heated vessel. The thermal-hydraulic flow in the vessel was simulated with the CFD code, and it in the loop was treated using the system code. Comparisons with experimental data were not presented; instead, only code-to-code comparisons were given. The results from such simple experiments could be expected to produce valuable data for verification and validation of the coupling method.

Session 13: PRIMARY CIRCUIT

Three papers have been presented in this session.

The first paper, presented by G. H. Lee, was concerned with a comparative study of the modelling of reactor internals for two reactor types: the PWR and PHWR. Test cases were presented of the 1/5 scaled APR+ model for the PWR and the moderator test vessel (Stern Laboratories Inc.) for the PHWR. The major conclusions can be summarized as follows. For PWR, the core-inlet-flow distribution could be predicted more accurately by considering the real geometry instead of a porous medium modelling approach to the internal structures located upstream of the core inlet. As a consequence of the complex geometry of the PWR internal structures, the magnitudes of the pressure drop in the axial direction differ locally depending on the radial distribution. Under the above-mentioned conditions, it appears that the Isotropic Loss Model (porous medium model) may have limited applicability.

For the PHWR, again an approach taking into consideration the real geometry of the tubes predicted local velocity and temperature distributions better than the porous medium model. However, in this case, the benefits were small compared to the extra computation cost. If licensing applicants use the porous medium model to predict PHWR moderator sub-cooling, care should be taken to validate the physical appropriateness and the conservativeness of the correlation for the pressure loss coefficients.

The second paper dealt with a loss-of-flow transient (LOFT) in a boron-neutron capture therapy (BNCT) converter, and was a novel addition to the papers usually presented at the CFD4NRS workshops. BNCT assumes that the source of therapeutic radiation is a nuclear reactor core. The neutrons coming directly from the core may not have the required parameters therapeutically, and for this reason fission converters are designed and utilized to modify the energy spectrum of the primary neutron beam, to make it useful for treatment purposes. The BNCT converter, due to its unique geometry and application location in the reactor core, was analysed by means of CFD simulation. This was considered as the best match technique to the scale of the problem to check all the thermal-hydraulic aspects and details of the flow pattern.

All the results prove the existence of sufficient safety margins even higher than expected in the analytical study for the most conservative approach. A critical flow rate for the "unlikely-to-occur" scenario of LOFT has been determined. The most important conclusion from the simulation can be stated as follows: according to the results obtained, during a LOFT, the flow would be reduced to 60% through the converter channel, the maximum temperature of water would not exceed 110˚C, even if the power of converter would be 2.5 times higher than in normal regime (conservative assumption). The evidence form the simulation leads to the conclusion that safe operation of the BNCT converter could be assured.

E. Laurien presented an "Experimental Investigation of Stratified Pipe Flow Mixing in a Horizontal T-Junction". However, since he was session chairman, he preferred not to offer a session summary for this paper.

Session 14: TWO-PHASE FLOW 2

The first paper in this session was presented by S. Mimouni. Here, a multi-field approach was employed in the modelling gas-liquid flows. Three fields or phases were used to represent the continuous liquid phase, the dispersed and continuous gas phases. The paper detailed the modelling of the drag forces between the phases. Mass transfer between the gas phases was considered to simulate the dispersion and merging of the gas bubbles. The validation examples provided demonstrate that the modelling approach can produce good results.

The paper was presented by J. Lehmkuhl. This paper considers the role of wall functions in the modelling of wall condensation. The paper presents a step-by-step approach in formulating new wall functions to include condensation mass transfer normal to the wall. Examples were provided to demonstrate the improvement of the new models over those usually employed, carried over from single-phase considerations.

The third paper was presented by T. Ma, describing his PhD thesis work in using LES to model bubbly flows. LES was used to model the turbulence in the liquid phase with the bubble-induced-turbulence (BIT) model of Sato to represent the effects of the bubble motions on the turbulence. The LES results were compared with those obtained using URANS, and against measured data. As expected, LES was able to resolve many more details of the flow, CFD predictions matching well with the measured data. This paper illustrates the advances made in extending LES to modelling multi-phase flows.

Session 15: TWO-PHASE FLOW 2

The first paper by Park et al. revisited an older problem, namely the penetration of a jet into a denser liquid pool (the case of a water jet into a corium pool) with a relatively new method: the moving particle semi-implicit (MPS) method, improved by the coupling to rigid-body dynamics (RD-MPS). The authors performed jet penetration calculations with various density ratios between the two fluids, and focused their comparisons with available experimental data concerning the jet penetration depth. The agreement was satisfactory in this respect, but there was no discussion of the capability of the method to predict properly the breakup of the jet, certainly important in the case of a vapour explosion.

The second paper by Mukin and Dehbi described experiments conducted at the ARTIST facility at PSI to study the flow on the secondary side of a steam generator in the case of a tube rupture; the important issue here is the scrubbing of aerosol particles that could be present in the primary fluid in case of core degradation. The ARTIST facility is a scaled portion of a steam generator. A tube having a circumferential opening was placed in the middle of the bundle to simulate the break. The gas escaping from the break was air. First attempts using the k-ε and Reynolds stress models showed that better CFD methods were needed, and so the improved delayed detached eddy simulation (IDDES) implemented into the open-source CFD code OpenFOAM was used to obtain much better agreement with the experimental data. The calculated flow fields qualitatively matched well the experimental data, and successfully reproduced the radial flow distribution patterns.

Session 16: OECD–PSI BENCHMARK WRAP-UP

This was essentially an open-forum question and answer (Q&A) session, primarily aimed at giving an opportunity for participants in the benchmark exercise to voice their opinions. No session summary was requested here.

ACKNOWLEDGEMENTS

Organizing Committee

Brian L. Smith, Paul Scherrer Institute, Switzerland, General Chair
Dominique Bestion, Commissariat à l'Energie Atomique, France, Co-Chair
Ghani Zigh, United States Nuclear Regulatory Commission, USA, Co-Chair
Horst-Michael Prasser, ETH Zurich, Switzerland, Head of Local Organizing Committee
Michele Andreani, Paul Scherrer Institute, Switzerland, Synthesis of the OECD-PSI Benchmark
Martin Kissane, OECD Nuclear Energy Agency, France, Technical Secretariat
Anthony Ulses, International Atomic Energy Agency, Austria, Observer

Scientific Committee

Michele Andreani, PSI, Switzerland
Arnoldo Badillo, PSI, Switzerland
Yann Bartosiewicz, UC Louvain, Belgium
Ulrich Bieder, CEA, France
Chris Boyd, US NRC, USA
Diana Caraghiaur, KTH, Sweden
Jose Luis Munoz Cobo, UPV, Spain
Abdel Dehbi, PSI, Switzerland
Horst Glaeser, GRS, Germany
Yassin Hassan, TMU, USA
Wang-Kee In, KAERI, S. Korea
J.W. Kim, SNUST, S. Korea
Bostjan Koncar, JSI, Slovenia
Djamel Lakehal, ASCOMP, Switzerland
Simon Lo, CD Adapco, UK
Annalisa Manera, UMICH, USA
Stephane Mimouni, EDF, France
Tadashi Morii, NRA, Japan
Victor Petrov, UMich, USA
Horst-Michael Prasser, ETHZ, Switzerland
Yohei Sato, PSI, Switzerland
Jean-Marie Seynhaeve, UC Louvin, Belgium
Brian L. Smith, PSI, Switzerland
Heikki Suikkanen, LUT, Finland
Timo Toppila, FORTUM, Finland
Kristian Angele, Vattenfall R&D, Sweden
Emilio Baglietto, MIT, USA
Sofiane Benhamadouche, EDF, France
Dominique Bestion, CEA, France
Sebastian Buchholz, GRS, Germany
Xu Cheng, KIT, Germany
Pierre Coste, CEA, France
Joerg Dreier, PSI, Switzerland
Estelle Graffard, AREVA, France
Thomas Hoehne, HZDR, Germany
Ralf Kapulla, PSI, Switzerland
Jordan Ko, AREVA, France
Eckhard Krepper, HZDR, Germany
Eckart Laurien, Stuttgart, Germany
Dirk Lucas, HZDR, Germany
Jean Marc Martinez, CEA, France
Fabio Moretti, UniPisa, Italy
Bojan Niceno, PSI, Switzerland
Markus Piro, AECL, Canada
Ferry Roelofs, NRG, NL
Martina Scheuerer, GRS, Germany
Medhat Sharabi, PSI. Switzerland
Chul-Hwa Song, KAERI, S. Korea
Iztoc Tiselj, JSI, Slovenia
Pascal Veber, ONSALA, Sweden

Tadashi Watanabe, JAEA, Japan
Jin Yan, GE, USA
Han-Young Yoon, KAERI, S. Korea
George Yadigaroglu, ASCOMP, Switzerland
Guan Yeoh, UNSW, Australia

Local Organizing Committee (ETHZ, Switzerland)

Horst-Michael Prasser
Christian Bolesch
John Kickhofel
Nora Kerenyi
Paolo D'Aleo
Nathan Lafferty
Robert Adams
Ralph Eismann
Abhishek Saxena