Unclassified **NEA/CSNI/R(2011)14**

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

23-Jan-2012

English text only

NUCLEAR ENERGY AGENCY
COMMITTEE ON THE SAFETY OF NUCLEAR INSTALLATIONS

NEA/CSNI/R(2011)14
Unclassified

Cancels & replaces the same document of 20 January 2012

Proceedings of the CFD for Nuclear Reactor Safety Applications (CFD4NRS-3) Workshop

September 14-16, 2010
Bethesda North Marriott Hotel & Conference Centre, Bethesda, MD, USA

English text only

JT03314725

Document complet disponible sur OLIS dans son format d'origine
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 30 OECD member 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 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 INSTALLATION

"The Committee on the Safety of Nuclear Installations (CSNI) shall be responsible for the activities of the Agency that support maintaining and advancing the scientific and technical knowledge base of the safety of nuclear installations, with the aim of implementing the NEA Strategic Plan for 2011-2016 and the Joint CSNI/CNRA Strategic Plan and Mandates for 2011-2016 in its field of competence.

The Committee shall constitute a forum for the exchange of technical information and for collaboration between organisations, which can contribute, from their respective backgrounds in research, development and engineering, to its activities. It shall have regard to the exchange of information between member countries and safety R&D programmes of various sizes in order to keep all member countries involved in and abreast of developments in technical safety matters.

The Committee shall review the state of knowledge on important topics of nuclear safety science and techniques and of safety assessments, and ensure that operating experience is appropriately accounted for in its activities. It shall initiate and conduct programmes identified by these reviews and assessments in order to overcome discrepancies, develop improvements and reach consensus on technical issues of common interest. It shall promote the co-ordination of work in different member countries that serve to maintain and enhance competence in nuclear safety matters, including the establishment of joint undertakings, and shall assist in the feedback of the results to participating organisations. The Committee shall ensure that valuable end-products of the technical reviews and analyses are produced and available to members in a timely manner.

The Committee shall focus primarily on the safety aspects of existing power reactors, other nuclear installations and the construction of new power reactors; it shall also consider the safety implications of scientific and technical developments of future reactor designs.

The Committee shall organise its own activities. Furthermore, it shall examine any other matters referred to it by the Steering Committee. It may sponsor specialist meetings and technical working groups to further its objectives. In implementing its programme the Committee shall establish co-operative mechanisms with the Committee on Nuclear Regulatory Activities in order to work with that Committee on matters of common interest, avoiding unnecessary duplications.

The Committee shall also co-operate with the Committee on Radiation Protection and Public Health, the Radioactive Waste Management Committee, the Committee for Technical and Economic Studies on Nuclear Energy Development and the Fuel Cycle and the Nuclear Science Committee on matters of common interest."

EXECUTIVE SUMMARY

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. As a matter of 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.

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 19 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, five 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. This confirms 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 long-term 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.

TABLE OF CONTENTS OF THE PROCEEDINGS

Poster Paper Session 3

Multiscale Computer Simulation of Fission Gas Discharge During Loss-of-Flow Accident in Sodium Fast Reactor, *I.A. Bolotnov, F. Behafarid, D.R. Shaver, S.P. Antal, K.E. Jansen, R. Samulyak, H. Wei and M.Z. Podowski*
Experimental Measurements of Droplet Size and Velocity Distributions at the Outlet of a Pressurized Water Reactor Containment Swirling Spray Nozzle, *A. Foissac, J. Malet, R.M. Vetrano, J.-M. Buchlin, S. Mimouni, F. Feuillebois, O. Simonin*

CFD Modeling of Wall Steam Condensation: Two Phase Flow Approach Versus Homogeneous Flow Approach, *S. Mimouni, N. Mechitoua, A. Foissac, M. Hassanaly, M. Ouraou*

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, *Adrian Tentner, Simon Lo, David Pointer, Andrew Splawski*

Poster Paper Session 4

The Role of Direct Simulations in Validation and Verification, *G. Tryggvason, J. Buongiorno*

A Coupled CFD Finite Element Analysis Methodology in a Bifurcation Pipe in a Nuclear Plant Heat Exchanger, *J.A. Dixon, A. Guijarro Valencia, P. Ireland, P. Ridland, N. Hills*

Interpreting Thermocouple Reading in Fuel Assembly Head – A CFD Studyy on Coolant Mixing, *Karoliina Myllymäki, Timo Toppila, Tellervo Brandt*

Effective Approaches to Simulation of Thermal Stratification and Mixing in a Pressure Suppression Pool, *Hua Li, Pavel Kudinov*

A Synergistic Use of CFD, Experiments and Effective Convectivity Model to Reduce Uncertainty in BWR Severe Accident Analysis, *Chi-Thanh TRAN, Pavel Kudinov*

Poster Paper Session 5

2D Simulation of Two-Phase Flow across a Tube Bundle with Neptune_CFD Code, *D. Soussan, S. Pascal Ribot, M. Grandotto*

CFD Modelling of the Test 25 of the Panda Experiment, *N. Mechitoua, S. Mimouni, M. Ouraou, E. Moreau*

On the Prediction of Boron Dilution Using the CMFD Code Transat: the Rocom Test Case, *M. Labois, J. Panyasantisuk, T. Höhne, S. Kliem, D. Lakehal*

Program Book

Technical Papers

Session 1 – Advanced Reactors (1)
CFD Analysis of Decay Heat Removal Scenarios of the Lead Cooled ELSY Reactor, *Michael Böttcher*

Evaluation of an Experimental Data Set to Be Validation Data for CFD for a VHTR, *Richard W. Johnson*

Lead Pressure Loss in the Heat Exchanger of the ELSY Fast Lead-Cooled Reactor by CFD Approach, *Alexandra Onea, Michael Böttcher, Dankward Struwe*

CDF Calculations of Wire Wrapped Fuel Bundles: Modelling and Validation Strategies, *Ulrich Bieder, Valérie Barthel, Frederic Ducros, Patrick Quéméré, Simone Vandroux*

Session 2 – Containment (1)

Validation of a Simple Condensation Model for Simulation of Gas Distributions in Containments with CFX, B. Schramm, *J. Stewering, M. Sonnenkalb*

Assessment of the Gasflow Spray Model Based on the Calculations of the Tosqan Experiments 101 and 113, *M.A. Mohaved, J.R. Travis*

CFD Modelling of Condensation of Vapour in the Pressurized Ppoolex Facility, *T.J.H. Pättikangas, J. Niemi, J. Laine, M. Puustinen, H. Purhonen*

Investigation of the Turbulent Mass Transport during the Mixing of a Stable Stratification with a Free Jet Using CFD-Methods, *Armin Zirkel, Eckart Laurien*

Session 3 – Boiling Flow (1)

Modeling of Turbulent Transport Term of Interfacial Area Concentration in Gas-Liquid Two-Phase Flow, *Isao Kataoka, Kenji Yoshida, Manasori Naitoh, Hidetoshi Okada, Tadashi Mori*

Applicability of Two-Phase CFD to Nuclear Reactor Thermalhydraulics and Elaboration of Best Practice Guidelines, *D. Bestion*

Numerical Simulation of Condensation in Bubbly Flow, *P. Ruyer, K. Keshk, F. Deffayet, Ch. Morel, J. Pouvreau, F. François*

Validation of Neptune_CFD 1.0.8 for Adiabatic Bubbly Flow and Boiling Flow, *A. Douce, S. Mimouni, M. Guingo, C. Morel, J. Laviéville, C. Baudry*

Session 4 – Bundle Flow

Experimental Benchmark Data for PWR Rod Bundle with Spacer-Grids, *Elvis E. Dominguez-Ontiveros, Yassin A. Hassan, Michael E. Conner, Zeses Karoutas*

CFD Analysis of the Matis-H Experiments on the Turbulent Flow Structures in a 5x5 Rod Bundle with Mixing Devices, *Hyung Seok Kang, Seok Kyu Chang, Chul-Hwa Song*

CFD Prediction of Pressure Drop for the Inlet Region of a PWR Fuel Assembly, *Jin Yan, Kun Yuan, Emre Tatli, David Huegel, Zeses Karoutas*

Numerical Simulation of the Flow in Wire-Wrapped Pin Bundles: Effect of Pin-Wire Contact Modeling, *E. Merzari, W.D. Pointer, J.G. Smith*

Session 5 - Fire

Recommendation for Maximum Allowable Mesh Size for Plant Combustion Analyses with CFD Codes, *M.A. Mohaved*

Validation Process of the Isis CFD Software for Fire Simulation, *C. Lapuerta, F. Babik, S. Suard, L. Rigollet*

CFD Analysis of a Hydrogen Explosion Test with High Ignition Energy in Open Space, *Hyung Seok Kang, Sang Baik Kim, Min-Hwan Kim, Hee Cheon No*

Session 6 – Dry Cask

Validation of Computational Fluid Dynamics Code Models for Used Fuel Dry Storage Systems, *Gregory Banken, Kamran Tavassoli, Jayant Bondre*

A 2-D Test Problem for CFD Modeling Heat Transfer in Spent Fuel Transfer Cask Neutron Shields, *G. Zigh, J. Jolis, J.A. Fort*

Pressure Drop Measurement of Laminar Air Flow in Prototypic BWR and PWR Fuel Assemblies, *E. Lindgren, S. Durbin*

Computational Fluid Dynamics Modeling Approach to Evaluate VSC-17 Dry Storage Cask Thermal Designs, *Kaushik Das, Debashis Basu, Jorge Solis, Ghani Zigh*

Validation of the FLUENT CFD Computer Program by Thermal Testing of a Full Scale Double-Walled Prototype Canister for Storing Chernobyl Spent Fuel, *Indresh Rampall, Kalyan K. Niyogi, Debu Mitra-Majumdar*

Session 7 – Advanced Reactors

Fluid Flow and Heat Transfer Analysis of the VHTR Lower Plenum Using the Fuego CFD Code, *Sal B. Rodriguez, Stefan Domino, Mohamed S. El-Genk*

Experimental Efforts for Predictive Computational Fluid Dynamics Validation, *J.R. Buchanan, Jr. and R.C. Bauer*

Particle Deposition on an Array of Spheres Using RANS-RSM Coupled to a Lagrangian Random Walk, *A. Dehbi, S. Martin*

Unsteady Validation Metrics for CFD in a Cylinder Array, *Brandon Wilson, Jeff Harris, Barton Smith, Robert Spall*

Session 8 – Boiling Flow (2)

Prediction of a Subcooled Boiling Flow with Mechanistic Wall Boiling and Bubble Size Models, *B.J. Yun, A. Splawski, S. Lo, C.-H. Song*

Assessment of the Heat Transfer Model and Turbulent Wall Functions for Two Fluid CFD Simulations of Subcooled and Saturated Boiling, *D. Prabhudharwadkar, M. Lopez de Bertodano, J. Buchanan Jr.*

Prediction of a Subcooled Boiling Flow with Mechanistic Wall Boiling and Bubble Size Models, *B.J. Yun, A. Splawski, S. Lo, C.-H. Song*

CFD Simulation of Critical Heat Flux in a Tube, *L. Vyskocil, J. Macek*

Use of Synchronized, Infrared Thermometry and High-Speed Video for Generation of Space- and Time-Resolved High-Quality Data on Boiling Heat Transfer, *Craig Gerardi, Hyungdae Kim, Jacopo Buongiorno*

Session 9 – Mixing Flow (1)

CFD Simulations of the Flow Mixing in the Lower Plenum of PWR's, *G. Pochet, M. Haedens, C.R. Schneidesch, D. Léonard*

Direct Steam Condensation Modeling for a Passive PWR Safety System, *D.R. Shaver, S.P. Antal, M.Z. Podowski, D.H. Kim*

Study of Thermal Stratification and Mixing Using PIV, *B. Yamaji, R. Szijártó, A. Aszódi*

Challenges for the Extension of Limited Experimental Data to Full-Scale Severe Accident Conditions Using CFD, *C. Boyd, K. Armstrong*

Session 10 – Plant Applications

Three-Dimensional Porous Media Model of a Horizontal Steam Generator, *T.J.H. Pättikangas, J. Niemi, V. Hovi, T. Toppila, T. Rämä*

Fibre Agglomerate Transport in a Horizontal Flow, *G.M. Cartland Glover, E. Krepper, H. Kryk, F.-P. Weiss, S. Renger, A. Seelinger, F. Zacharias, A. Kratzsch, S. Alt, W. Kästner*

LES with Acoustics and FSI for Deforming Plates in Gas Flow, *Per Nilsson, Eric Lillberg, Niklas Wikström*

CFD Calculation of the Pressure Drop Through a Rupture Disk, *L. Mengali, D. Melideo, F. Moretti, F. D'Auria, O. Mazzantini*

A Shallow Water Equation Solver and Particle Tracking Method to Evaluate the Debris Transport, *Young Seok Bang, Gil Soo Lee, Sweng-Woong Woo*

Session 11 – PTS

Pre-Test CFD Simulations on Topflow-PTS Experiments with ANSYS CFX 12.0, *P. Apanasevich, D. Lucas, T. Höhne*

Transient Computational Fluid Dynamics Analysis of Emergency Core Cooling Injection at Natural Circulation Conditions, *Martina Scheuerer, Johannes Weis*

Validation of the Large Interface Method of Neptune_CFD 1.0.8 for Pressurized Thermal Shock (PTS) Applications, *P. Coste, J. Laviéville, J. Pouvreau, C. Baudry, M. Guingo, A. Douce*

PTS Prediction Using the CMFD Code TransAT: the COSI Test Case, *M. Labois, D. Lakehal*

Session 12 – Containment (2)

Validation of CFD-Models for Natural Convection, Heat Transfer and Turbulence Phenomena, *J. Stewering, B. Schramm, M. Sonnenkalb*

Toward a CFD-Grade Database Addressing LWR Containment Phenomena, *Domenico Paladino, Michele Andreani, Robert Zboray, Jörg Dreier*

Interaction of a Light Gas Stratified Layer with an Air Jet Coming from Below: Large Scale Experiments and Scaling Issues, *E. Studer, J. Brinster, I. Tkatschenko, G. Mignot, D. Paladino, M. Andreani*

Hydrogen Deflagration Simulations under Typical Containment Conditions for Nuclear Safety, *J. Yáñez, A. Kotchourko, A. Lelyakin*

Session 13 – Boiling Flow (3)

Experimental Data on Steam Bubble Condensation in Poly-Dispersed Upward Vertical Pipe Flow, *D. Lucas, M. Beyer, L. Szalinski*

Coupled Lagrangian and Eulerian Simulation of Bubbly Flows in Vertical Pipes: Validation with Experimental Data Using Multi-Sensor Conductivity Probes and Laser Doppler Anemometry, *José L. Muñoz-Cobo, Sergio Chiva, Santos Mendes, Mohamed Ali Abdelaziz*

Prediction of Polydisperse Steam Bubble Condensation in Sub-Cooled Water Using the Inhomogeneous Musig Model, *C. Lifante, T. Frank, A.D. Burns, D. Lucas, E. Krepper*

Session 14 – Mixing Flow (2)

Large Eddy Simulation of a Turbulent Flow in a T-Junction, *Jungwoo Kim, Jae Jun Jeong*

Numerical Simulations of Thermal-Mixing in T-Junction Piping System Using Large Eddy Simulation Approach, *Masaaki Tanaka, Hiroyuki Ohshima*

Large Eddy Simulations for Thermal Fatigue Predictions in a T-Junction: Wall-Function or Wall-Resolve Based LES, *S.T. Jayaraju, E.M.J. Komen, E. Baglietto*

Towards Empiricism-Free Large Eddy Simulation for Thermo-Hydraulic Problems, *V.M. Goloviznin, S.A. Karabasov, M.A. Zaitsev*

Dispersion of Radionuclides and Radiological Dose Computation over a Mesoscale Domain Using Weather Forecast and CFD Model, *R.B. Oza, V.D. Puranik, H.S. Kushwaha, Krishna Prasad, Arun Murthy*

TECHNICAL SESSION SUMMARIES

Session 1: Advanced Reactors (1) (Session Co-Chairs: Y. Hassan and V. Mousseau)

The first talk concerned CFD modeling of the primary circuit of a lead-cooled fast reactor. One of the modeling issues emphasized was the solidification of the lead coolant due to freezing. The talk illuminated the competing goals of ensuring a mesh-converged solution and the importance of doing model sensitivity studies. Mesh convergence is often the largest burden on the computer resources available. The mesh-converged runs can take weeks to complete. Because of the long run time, and sensitivity studies are often impractical.

The second talk was about validation of CFD for the lower plenum of a high temperature gas reactor. This talk pointed out the difficulties of not having CFD people involved in the original design of the experiment. Experimentalist and CFD people need to work together to ensure that the correct measurements are made at a quality level to make them appropriate for CFD validation. One of the successes of this work was to measure mass flow rate by two independent techniques. The hardest part of this work was addressing the computational cost of multi-time-scale transient simulations. In a simulation that has important fast (i.e. high frequency) information as well as important slow (low frequency) information, the time step has to be chosen small enough to resolve the high-frequency physics and the transient has to be run long enough to resolve the low-frequency information. This can cause prohibitive computer run times.

The third talk discussed using CFD to produce a conservative estimate of the pressure drop in a steam generator of a lead-cooled fast reactor. The approach here was to run the simulation with different turbulence models to find the one that resulted in the largest pressure drop. This is in contrast to other work where CFD was employed in a best-estimate plus uncertainty mode.

The fourth talk was on CFD modelling of the core of a sodium-cooled fast reactor with wire-wrapped fuel pins. The talk emphasized that resolving the wire wrap on the fuel pins requires a prohibitively large mesh. Therefore the mesh generation was a compromise between accuracy and acceptable computer cost. The validation of the CFD code was undertaken with 30 year old experimental data.
Summarizing, the two points that came out in all of the talks were the large computational cost and the difficulty of finding good CFD validation data. Although everyone wanted to get a mesh converged CFD solution, the complexity of nuclear reactor safety calculations often makes this impractical. The result is that often “mesh converged” means the largest grid that can effectively be computed. The second point was the difficulty in finding high quality CFD validation data for nuclear reactor safety applications. In this field, experiments have been designed to validate system codes, and new experiments are needed with the fidelity to validate CFD codes.

Session 2: Containment (1) (Session Co-Chairs: Jay Sanyal and Sam Durbin)

This session focused on the application of CFD in modelling severe accident scenarios in various reactor systems. There were four presentations in this session, three from academic/research organizations and one from a corporation. Overall, all the presentations had detailed descriptions of the problem at hand and were able to clearly articulate the need to use CFD as an important predictive tool in mitigating catastrophic failures. The presentations also involved comparison and validation of CFD results through carefully constructed and published experimental data. Following are the details of the individual presentations.

In the first, a custom subroutine was developed for CFX to model bulk condensation of steam, as an alternative to the default models in the code, which are significantly more computationally expensive. The CFD model was described in detail in the presentation and a discussion of the sensitivity analyses performed to obtain grid-independent results. The presentation also included detailed error estimates for both the CFD and experimental data, and thus was able to clearly demonstrate the feasibility as well as the limitations of conducting complex two-phase simulations in a containment region. The CFD results were compared with well-known PANDA experiments, and the authors included best practice guidelines for the CFD simulations. As a point of criticism, the author should have included a more detailed description of the specific condensation model used and especially the justification for the choice of parameters.

In the second presentation, the author presented a CFD simulation and validation data of spray injection using the GASFLOW FVM code (originally developed at LANL). The results were validated against EPR, MISTRA and TOSQAN spray model experiments. The author spent an inordinately long time describing the experimental set-up and not enough time describing the CFD model. Typically, spray-models are Euler/Lagrange and perhaps the author presumed that the audience was familiar with the particular numerical model used, though a more detailed and complete description of this should have been presented. Also, although the author presented a detailed comparison with experimental results, most of the plots included steady-state regions where there was no significant variation of the parameters of interest. Perhaps the author should have focused more closely on regions of rapid initial variation, and talked about discrepancies between predicted and observed results. Also, sprays are frequently characterized by break-up and coalescence, and it was not clear from the presentation whether either one of these processes had been modeled or not.

In the third presentation, the author presented a FLUENT Euler/Euler simulation with a custom condensation model in the containment of a BWR. The model was explained in detail with all equations and relevant assumptions clearly illustrated. The CFD results were compared with data from the PPOOLEX experimental facility. The authors ran a fully 3D transient, two phase simulation with multiple species in the gas phase. The authors also adhered to best practice guidelines for CFD calculations, especially in demonstrating grid-independent solutions. The comparisons to experiments were however qualitative in nature, which is possibly due to the fact that it is virtually impossible to obtain experimental data which is of the level of detail required for validating a CFD simulation in this context. Additionally, the authors used standard k-epsilon models for turbulence which are diffusive in nature and could explain some of the discrepancies in comparison to experimental observations.

In the fourth presentation, the author presented a turbulence model to simulate turbulent mass transfer involving mixing of a stable stratified flow and a free jet. The model is called the Turbulent Scalar Flux model. The model involved several adjustable parameters which are being currently investigated for stratification of a free jet. The model is similar in flavor to the RSM and is being validated against the THAI experiments. The challenge for the author would be to extend this model for multiphase flows which are most frequently observed in nuclear reactors. It seems the applicability of the model so far is limited to the specific cases it has been developed for, and it remains to be seen how it compares to other well-known and tested models such as LES or RSM. The application seems so far to be an academic exercise. There was no application of best practice guidelines.

Session 3 Boiling Flow (1) (Session Co-Chairs: Emilio Baglietto & Dirk Lucas)

This session focused on multiphase application and Best Practice Guidelines (BPGs) as they apply to nuclear reactor safety. The presentations and discussions were very valuable. Among the four presentations, two works presented validation of two-fluid models, one work presented a different flavour of modelling interfacial area concentration in two-fluid modelling, and the last work presented an interesting summary of the WG3 document of the OECD-CSNI-GAMA activity aimed particularly at proposing a methodology to consistently select model options and possibly a standardized nomenclature for the assembled modelling methods.

The first presentation was about modelling of turbulent transport term of inertial area concentration in gas-liquid two-phase flow. The second presentation dealt with application of two-phase CFD to nuclear reactor thermal hydraulics and elaboration of best practice guidelines. The third presentation used CFD to predict condensation in boiling flow. The fourth presentation was about validation of Neptune_CFD 10.8 for adiabatic bubbly flow and boiling flow.

One presenter has proposed a useful simulation tree to help verify the consistency of the assembled two-phase model. There is a large consensus on these recommendations and on the fact that one cannot randomly mix and match the various components. At the same time, there seems to be general agreement that while the consistency is a necessary condition, it is not a sufficient one, and the adequacy or quality of the submodels is the critical node. There are a wide variety of closure models available, and more coming (as one of the papers introduced), and clearly the greatest emphasis is presently being placed on testing these closures and learning their capabilities and limitations. One often refers to this testing as validation, but, for the example presented, work should probably be referred to as testing rather than validation. There are attempts to adopt some BPGs, but still the format is not that of a rigorous validation procedure.

It is agreed that in order to construct more general models, separate-effect tests are extremely useful. These tests are neither readily available nor simple to construct, and while we all wish to have better data we must learn how to best deal with the available data and possibly supplement them with first-principles numerical simulations.

One interesting conclusion is that although we clearly have space for improving and generalizing the closure models, and hundreds of coefficients within them, the testing gives clear evidence that well assembled two-fluid models with sufficient physical representation show great potential and often results of very high quality.

Session 4: Bundle Flow (Session Co-Chairs: Walter Schwarz and Brian Smith)

This session had four presentations focusing on different aspects of flows through a rod bundle. The first was about experimental data from a 5x5 test section representative of a PWR bundle with mixing vanes. The second presentation dealt with experiments and CFD on a 5x5 test section with mixing vanes. The third presentation used CFD to predict the flow behavior at the inlet nozzle of a 17x17 PWR rod bundle. The fourth presentation was a CFD study of wire-wrapped fuel elements in a sodium cooled reactor.

Common points from the presentations were:

- There is a need to model the geometry as precisely as possible to capture the details of the flow, and often simplification is difficult to justify.
- There is interest in being able to predict the pressure drop through different parts of a fuel assembly.
- There was a recognition and adherence to Best Practice Guidelines (BPGs) for CFD:
    - Variation of meshing to obtain mesh-independent solutions;
    - Variation of choice of turbulence models to capture the flow physics.

The key take-away points from this session are:

- Experiments can provide insight into the physics of the flow development in bundles;
- CFD is now recognized as an accepted analysis tool;
- Attention to geometric detail is important;
- It is important to follow BPGs;
- Most of the presentations reported work which utilized commercial CFD software, but one employed an in-house spectral element code.

Session 5: Fire (Session Co-Chairs: John Mahaffy and Jason Dreisbach)

The Fire session included three papers, ranging from a V&V application for a new, open source CFD code, to application of CFD to hydrogen explosion, and guidance for mesh size for fluid-structure interactions. All papers were well received.

In the first presentation, an open-source CFD code called ISIS was used and validated using a ventilated compartment fire. The tiered approach to validation appears to pay off well as the code predicted the subject parameters within experimental error. Questions from the audience focused on the mechanics of the code itself, and comparison with other codes used for similar purposes.

Some interesting thoughts to consider:
Example validation was for one experiment. Are other comparisons as good? There was no quantitative measure of the accuracy of fit between experiment and prediction. Rather, classic qualitative validation judgment was used. More should be done to develop a quantitative measure of accuracy of code predictions. The code was developed as open source. Are others in the community using open source CFD? Are others developing open source CFD?

The second presentation was about CFD modeling of a hydrogen explosion test series that used a range of conditions and barriers. The goal was to evaluate the appropriate stand-off distance between hydrogen generating plant and nuclear power plant using CFD. Based on the comparison between code and experiment, a modified reaction rate was proposed to more accurately represent the near-field pressure wave. Evaluation of far-field pressure dissipation was not discussed.
The third presentation provided a novel idea for selecting maximum mesh sizes for CFD calculations. The particular CFD application was for hydrogen explosion effects on containment structures. The suggested approach for mesh size generation identified the need to couple the fluid and structural calculations. A theoretical evaluation yielded a mesh size dependence on the speed of sound in gas and structural response frequency. The concept is simple and would be appropriate for inclusion in Best Practice Guidelines.

Session 6: Dry Cask (Session Co-Chairs: Jorge Solis and Chris Bajwa)

This session included four papers discussing the application and validation of CFD codes for dry cask storage and transportation.

The first paper dealt with methodology, and steps were developed to achieve validation of the Fluent CFD code against test results for one of Transnuclear's high capacity storage systems. The steps included the choice of meshing scheme, fixed vs. temperature dependent thermal properties, choice of turbulence modelling approach (including wall treatment), and the use of steady vs. transient mode modelling techniques to capture the turbulent nature of the flow and heat transfer. The work was not original and did not attempt to use existing CFD Best Practice Guidelines.

The second presentation was concerned with spent fuel transfer casks, including validation of the Fluent and StarCCM+ CFD codes for such applications. These transfer casks are horizontal cylindrical steel enclosures with integral gamma and neutron radiation shields. The studied geometry is a 5 cm annulus and a diameter of 2 m. A wide range of spatial and temporal scales makes this problem challenging to solve using CFD. Relevant experimental data at these scales are not available in the literature, but some recent modelling studies offer insights into numerical issues and solutions. The geometries in these studies, and experimental data in the literature at smaller scales, all have large annular gaps that are not prototypic of the transfer cask neutron shield. The work is original and the results in the paper can be used to provide input to the CFD BPGs for this application. Due to the critical nature of the application, the argument is made for new experiments at representative scales.

The third presentation was about pressure drop experimental measurements through two commercial fuel assemblies, including 17x17 PWR and 9x9 BWR bundles. The assemblies were tested in the laminar regime with Reynolds numbers ranging from 10 to 1000, based on the average assembly velocity and hydraulic diameter. Pressure-drop measurements were made across individual bundle spans and grid spacers in the mock fuel assemblies using high-sensitivity differential pressure gauges. The work is original and these type of data can be used in validation for applications involving evaluation of the complete loss-of-coolant accident scenario in spent fuel pools as well as performance analysis of dry storage casks.

The fourth presentation dealt with CFD validation of the ventilated concrete storage cask VSC-17 system. In this application, the Fluent software was used. Turbulence model sensitivity analyses was undertaken in this work. The sensitivity involved the standard k-ε model, RNG k-ε model and the k-ω turbulence model. The turbulence model sensitivity study included also the effect of wall treatment model choice. Sensitivity to boundary conditions was also examined. The work is not original, but the experimental data used is a good source for CFD validation.

Session 7: Advanced Reactors (2) (Session Co-Chairs: Morii Tadashi and Jay Sanyal)

This session focused exclusively on experimental validation of CFD modelling. Specifically, the authors emphasized building "bottom up" unit-level experiments that could be used to a) quantify errors leading up to the complete experimental set-up, and b) to provide a suitable basis for comparison with CFD models in terms of initial and boundary conditions. The authors proposed methods for systematically quantifying errors in both experiments and CFD calculations**.** Although most of the results were for single phase-flow, and are valuable in their own right, it is somewhat baffling that none of the studies investigated multiphase flows, which are frequently observed in nuclear reactors.

In the first presentation, the author presented an application of a general-purpose, three-dimensional FV code called FUEGO, developed at Sandia. The code was validated against experimental data on swirling jets and then applied to an LP plenum with multiple jet interactions and stationary structures. According to the author, about $50 million was spent in developing these capabilities, although it would appear that any commercial CFD code would be able to easily model the presented scenarios. Nevertheless, the author presented a rigorous way of validating CFD codes through carefully constructed experiments.

In the second presentation, the author presented P-CFD, a consistent set of input that produces output data for which the uncertainty can be accurately characterized. Basically, this holds experimental results to the same level of scrutiny that computations are frequently subjected to, and also provides a consistent basis for comparison between the two. The essence of the method lies in devising unit-level, well-understood experimental cases that can be used to validate CFD results. The motivation behind this bottom-up approach is the need to be able to accurately quantify the errors associated with both experimentation and simulation. The author pointed out that experiments and numerical analyses complement each other, so there needs to be a high level of cooperation between them for a successful V & V process.

In the third presentation, the author presented an Euler/Lagrangian simulation of particle deposition on arrays of spheres. The approach is not novel and based on standard Lagrangian particle tracking with stochastic dispersion. The author was presenting on behalf of a colleague and did not have access to all details of the simulation. The study was conducted for dilute particle loading and no attempt to include particle/particle interaction was made. The particles were considered to "stick" to walls upon impact. Although lacking in originality, the study did validate the flow and turbulence models for standard flow past spheres before adding particle dispersion to the simulations.

In the fourth presentation, the author reiterated the need to formulate a consistent and systematic basis for validation of CFD much along the lines proposed by Jack Buchanan. He presented experimental results on flow past cylinder arrays and compared them to numerical data produced using different turbulence models, the Detached Eddy Simulation model performing the best. The essence of the talk was a detailed comparison of unsteady experimental data against CFD simulations which mirrored the exact boundary and initial conditions.

Session 8: Boiling Flow (2) (Session Co-chairs: Dominique Bestion and Dirk Lucas)

All papers of this session fully complied with the scope of the workshop, and were relevant to nuclear reactor safety.

The three first papers dealt with the validation of CFD models for boiling flow, including some sensitivity to the available closure options. All CFD applications used the RANS approach. Attention was focused on wall transfers, including wall heat partitioning and DNB criterion. Reasonable predictions of experimental data were obtained, but the experimental information was not sufficient to provide a separate-effect validation of every closure model. For this reason, deficiencies of the single closure models may compensate each other. A first attempt to predict DNB with CFD using a very simple local DNB criterion produced an accuracy of about 10% in a simple geometry for a rather large range of flow rate, pressure and diameter. A more accurate and physically based criterion would need more experimental information on the local flow processes which are responsible for the DNB occurrence. The three papers give a good idea of the present state of the art in using CFD for boiling flow and DNB investigations.
There was some application of BPGs reported in all three papers, in particular to check the independence of results to mesh size.

The fourth paper presented advanced experimental techniques used in boiling flow up to DNB. This is exactly the type of experiment which may provide the information required to progress in the CFD modelling. These experiments were limited to pool boiling, and one can encourage further efforts to extend the investigations to convective boiling.

One must keep in mind that RANS simulations are very far from direct simulations and many closure laws are required which need experimental data for separate-effect validation. The future progress of the CFD RANS approach used for boiling flow will mainly depend on the new experimental data which will be made available.

Session 9: Mixing Flow (1) (Session Co-Chairs: Walter Schwarz and John Mahaffy)

This session had four presentations on flow mixing analyses. These presentations included:

- CFD of mixing in the lower plenum of a PWR, calculating mixing coefficients and flow distribution;
- Modelling and analysis of direct steam condensation in an advanced PWR;
- Experimental and CFD study of thermal stratification and mixing in a relatively simple geometry;
- CFD of severe accident natural circulation flows, and the challenges involved.

Common points from the presentations are as follows:

- All presentations involved code validation and two of them extended the modeling to full-scale reactor scenarios;
- All presentations included the effects of grid sensitivity studies and turbulence model sensitivity studies, or at least scrutinized or rationalized the choices based on previous work.

The key take-away points from this session are:

- Good application of CFD doesn't necessarily buy increased safety margin, but helps to understand it;
- Collaboration between academia and industry is occurring and can produce valuable results;
- It is useful to keep a view of the physics when interpreting the adequacy of CFD predictions;
- Challenges abound in which one is faced with a requirement to simulate a full-scale reactor scenario, because there is often little relevant experimental data, and uncertainty in the boundary conditions and grid sensitivity study demands must be balanced against computational resources;
- When applying CFD to real problems, one should never lose sight of the "big picture" in guiding decision-making in the details of your CFD modelling approach

Session 10 Plant Applications (Session Co-Chairs: Emilio Baglietto & Yassin Hassan)

This session included five presentations. Among the papers, three were pure CFD applications, one involved solving shallow-water equations, and the last was about the application of a porous body approach. The papers involved a wide range of application and physics, including single phase, multiphase and Fluid Structure Interaction (FSI).

Plant applications included:

- Three-Dimensional Porous Media Model of a Horizontal Steam Generator
- Fiber Agglomerate Transport in a Horizontal Flow
- LES with Acoustics and FSI for Deforming Plates in Gas Flow
- CFD Calculation of the Pressure Drop through a Rupture Disk
- A Shallow Water Equation Solver and Particle Tracking Method to Evaluate the Debris Transport.

All topics are clearly related to nuclear reactor safety applications, with two papers on debris transport and the others on components operation and reliability. This session had a mix between method development, validation and application.

Trying to derive conclusions was not simple as the applications and methods were very sparse:

- On one side, it is interesting to note that from the application point of view there are still tendencies to use simpler approaches rather than full 3D CFD, in this case employing 2D shallow-water equations in one case, and the porous body approach in the other.
- In contrast, there were advanced applications using acoustic analysis and FSI with a fully coupled solution approach, and good efforts in evaluating its potential and limitations..

- It is clear that there are still quite different opinions on computational requirements and feasibility of applications.

- In this session, there was still limited use of BPGs in the applications. Sometimes, only a single computational grid with first-order spatial discretization was employed. This clearly limits the understanding from the applications, since physical and numerical errors are still superimposed.

Session 11: PTS (Session Co-Chairs: Eric Volpenhein and Chris Boyd)

Four works were presented in the Pressurized Thermal Shock session (PTS). It was observed that PTS remains a demanding application for CFD, combining some of the simulation challenges of T-junction turbulent mixing, two-phase stratified flow, and buoyancy-driven plumes. In addition to PTS, these works accordingly examine the use of CFD for characterizing reactor conditions of broad contemporary interest to the international reactor safety community.

The session provided information of noteworthy scientific value, generally building upon prior works to advance CFD technology and experience in incremental fashion. The sessions explored a variety of computational methods including single- and two-fluid models, Interface Tracking Methods, and hybrid approaches featuring Large Interface Tracking and Very Large Eddy Simulation (V-LES). Sensitivity analyses were commonly included to quantify the influence of turbulence models and mesh construction. Assessments of analysis results were also generally supported by a mix of high quality separate-effects and integral experimental data.

The presentations demonstrated virtually universal awareness and attention to BPGs, with varied success in implementation. Application of BPGs toward simulation of PTS was, in fact, the focus of one presentation, providing unique insight into the BPGs themselves. Mesh dependency and other sensitivities were most often investigated. Where substantial compliance to BPGs was not complete, the work was acknowledged as “work-in-progress”. In some cases, explicit qualification of conclusions pending more complete compliance with BPGs would be appropriate.

Regarding suitability for code validation, the works performed represented appropriate and substantial steps, though it was observed that more thorough and consistent comparison with included experimental data would be helpful. Furthermore, a deeper assessment of the model strengths and limitations with the reported models with regard to the underlying phenomena and numerical methods would improve confidence in the generality and predictive capabilities. Caution is also advisable when formulating conclusions in the absence of more complete compliance with BPGs.

Session 12 – Containment (2) (Session Co-chairs: Fabio Moretti and Dominique Bestion)

All papers of this session were fully complying with the scope of the workshop, and relevant to nuclear reactor safety.

The first dealt with the validation of a CFD model for phenomena that govern the distribution of hydrogen and steam in a containment during severe accident scenarios.

The second paper presented a selection of experimental tests from the SETH-PANDA program, and demonstrated the availability and adequacy of a CFD-grade experimental database for code validation for containment problems.

The third paper addressed the characterization and modelling of stratification build-up and stratification break-up phenomena in containment atmospheres, making reference to PANDA and MISTRA experiments.

The fourth paper presented CFD simulations of hydrogen deflagration, performed with a recently developed in-house code.

Regarding the papers focusing on experiments, CFD-grade data were presented and discussed, with particular attention to the test repeatability issue and, at least in one case, with an effort to evaluate the experimental uncertainties. At least two papers presented physical and theoretical analyses (with a noteworthy attempt to perform a scaling analysis), carried out prior to completing the modelling and the numerical simulation. Even if large computing capabilities are available, one should never forget to do this kind of analysis as a preliminary step in the methodology, which may then involve some application of CFD.

Session 13: Boiling Flow (3) (Session Co-chairs: B. L. Smith and Chul-Hwa Song)

This was a short session, of just three papers, but both the Boiling Bubbly Flow (1) and Boiling Bubbly Flow (2) sessions had four papers each, demonstrating that the topic was of more general interest than the number of presentations indicates. None of the papers described CFD approaches to simulate boiling from first principles, but rather were focused on using CFD to predict the bubble dynamics, with and without mass and energy exchange between the phases. From the viewpoint of presentation, the papers were nicely balanced, with new experiments being reported together with associated CFD analyses.

Paper 1 described interesting new data from the TOPFLOW facility at FZD Rossendorf in which steam is injected through jets into a vertical pipe (height 8m, diameter 200mm) containing sub-cooled water. Experiments were made for different flow conditions, varying the pressure, liquid flow rate, steam injection rate and degree of sub-cooling. Local measurements were made using wire-mesh sensors. Analysis of a selected test was described in Paper 3 in which ANSYS CFX was employed using a two-fluid Euler/Euler modelling approach with a recent extension of the inhomogeneous MUSIG model to allow for bubble condensation. Comparisons between measurement and calculation of the radial distribution of void fraction and bubble-size distributions showed good correspondence.

Paper 2 dealt with air-water flow, and centred on work being carried out at the University of Valencia, Spain. Tests were conducted using the PUMA experimental facility in which air is injected at the base of a water column and local measurements of void fraction, interfacial area concentration, interfacial velocity and Sauter mean diameter measured using a four-sensor conductivity probe. Liquid velocity and turbulence intensity were also measured using Laser Doppler Anemometry. Numerical simulation of the experiments was undertaken using an in-house code and employing a Lagrange/Euler modelling approach.

In summary, 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.

Session 14: Mixing Flow (2) (Session Co-Chairs: Chris Boyd and Fabio Moretti)

All of the papers in Session 14 were considered original works and were within the scope of the conference. Each paper was directly tied to nuclear reactor safety issues in some way.

The papers indicated a consideration of CFD Best Practice Guidelines, but there was no systematic approach to follow the guidelines.

Four of the five papers were related to LES simulation of a T-Junction problem and there were direct comparisons with the data. There was a discussion of the details of the fluid structures and interactions that went beyond the T-Junction problem to add further insights into the predicted phenomena. This type of discussion is beneficial to the modelling community.
Two of the papers discussed ways to reduce the computational requirements for the T-junction models through investigations of simplified modelling approaches. One paper demonstrated the adequacy of using a wall-modelling approach compared to full resolution of the wall layer. This type of approach is important for analysts considering high Reynolds number plant simulations. An interesting Reynolds scaling approach was employed in one paper. In another, a low-dissipative and low-dispersive CABARET scheme was applied to the T-junction problem and relatively decent results were produced with a relatively small and efficient computational model.

In a fifth paper, atmospheric dispersion modelling was discussed. Such simulations are noted to be challenging, due to the large extent of the model domain (on the order of kilometers) and the variability of the boundary conditions (incorporating weather conditions). It is noted that this is a “work-in-progress” endeavour, and that further refinements and consideration of best practices are needed.