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

14-Feb-2014

English - Or. English

NUCLEAR ENERGY AGENCY
COMMITTEE ON THE SAFETY OF NUCLEAR INSTALLATIONS

CFD for Nuclear Reactor Safety Applications (CFD4NRS-4)

Workshop Proceedings

10-12 September 2012
Daejeon Convention Centre (DCC)
Yuseong, Daejeon, Korea

This paper only provides the list of presentations and the summary of the technical sessions.

The full presentations or papers are available only in electronic form.

JT03352480

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.

This work is published on the responsibility of the OECD Secretary-General.

NUCLEAR ENERGY AGENCY

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

The mission of the NEA is:

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

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

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

THE COMMITTEE ON THE SAFETY OF NUCLEAR INSTALLATIONS

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

The last decade has seen an increasing use of three-dimensional Computational Fluid Dynamics (CFD) and Computational Multi-Fluid Dynamics (CMFD) codes in predicting single-phase and multi-phase flows under steady-state or transient conditions in nuclear reactors. The reason for the increased use of multi-dimensional CFD methods is that a number of important thermal-hydraulic phenomena cannot be predicted using traditional one-dimensional system analysis codes with the required accuracy and spatial resolution. CFD codes contain empirical models for simulating turbulence, heat transfer, multi-phase interaction and chemical reactions. Such models must be validated before they can be used with sufficient confidence in nuclear reactor safety (NRS) applications.

The necessary validation is performed by comparing model predictions against trustworthy data. However, reliable model assessment requires CFD simulations to be undertaken with full control over numerical errors and input uncertainties to avoid erroneous conclusions being drawn. These requirements have prompted an OECD/NEA initiative to form writing groups of experts with the specific task of assessing the maturity of CFD codes for NRS applications, and to establish a data base and Best Practice Guidelines (BPGs) for their validation.

Scope

Following the CFD4NRS workshops held in Garching, Germany (Sept. 2006), Grenoble, France (Sep. 2008) and Washington D.C., USA (Sept. 2010), this Workshop is intended to extend the forum created for numerical analysts and experimentalists to exchange information in the application of CFD and CMFD to NRS issues and in guiding nuclear reactor design thinking. The workshop includes single-phase and multi-phase CFD applications, and offers the opportunity to present new experimental data for CFD validation. Emphasis has been in the following areas:

- More emphasis has to be given to the experiments, especially on two-phase flow, for advanced CMFD modelling for which sophisticated measurement techniques are required.

- It is very important to deepen understanding of the physics before starting numerical analysis.

- Single-phase and multi-phase CFD simulations with a focus on validation are welcome in areas such as: single-phase heat transfer, boiling flows, free-surface flows, direct contact condensation and turbulent mixing. These should relate to NRS-relevant issues, such as pressurised thermal shock, critical heat flux, pool heat exchangers, boron dilution, hydrogen distribution in containments, thermal striping, etc. The use of systematic error quantification and the application of BPGs are strongly encouraged.

- Experiments providing data suitable for CFD or CMFD validation are also welcome. These should include local measurements using multi-sensor probes, laser-based techniques (LDV, PIV or LIF), hot-film/wire anemometry, imaging, or other advanced measuring techniques. Papers should include a discussion of measurement uncertainties.

Results and their significance

There were over 150 registered participants at the CFD4NRS-4 workshop. The programme consisted of 48 technical papers. Of these, 44 were presented orally and 4 as posters. An additional 8 posters related to the OECD/NEA–KAERI sponsored CFD benchmark exercise on turbulent mixing in a rod bundle with spacers (MATiS-H) were presented and a special session was allocated for 6 video presentations. In addition, five keynote lectures were given by distinguished experts.

The number of participants represents a 25% decrease with respect to the previous CFD4NRS-3 Workshop held in Washington DC in September 2010. Nonetheless, this attendance record compared favourably with the second Workshop in the series, XCFD4NRS, held in Grenoble in 2008, and a two-fold increase compared to the first Workshop, held in Garching in 2006. Factors influencing the slight fall in attendance are: (i) fewer domestic students; (ii) the NUTHOS-9 conference being held in Taiwan at exactly the same time; (iii) the expense involved in making the trip to Korea from Europe and (especially) the US; (iv) the negative impact on nuclear research following the Fukushima disaster in March 2011.

The papers given at the Workshop covered different nuclear safety topics, and, for the first time, some reactor-design issues. However, the ratio of papers devoted to experimentation to those devoted to analysis was not as well balanced as previously seen, with too few experimental studies reported. Progress in modelling, and improvements in the use of the Best Practice Guidelines for performing quality CFD computations can only result from pursuing a programme of analysis of a multitude of CFD-grade experiments. A wrong idea circulates, particularly among managers, that CFD simulations may ultimately replace costly experimentation. This is only partially true in the case of prototypic experiments, but CFD tools include many models and closure laws: these have to be properly validated, and this can only be achieved by means of experiments. It remains a primary objective of the CFD4NRS series of Workshops to bring together the experimenters providing the data needed to improve the physical models in CFD codes with the analysts who utilise these models.

Switzerland is a candidate to host the next Workshop in 2014 and will be organised by staff at the Paul Scherrer Institute (PSI); PSI has also volunteered to sponsor and organise the third OECD/NEA CFD benchmark exercise based on an experiment to be performed in the containment test facility PANDA. In the panel session at the close of this 4th Workshop, delegates confirmed their interest in attending a follow-up Workshop, and considered the two-year interval to be appropriate.

As is customary at the panel session, which in this case was led by B. L. Smith (PSI) and D. Bestion (CEA), summaries were made by the respective session chairpersons of the presentations that were given during the 12 oral sessions, and comments invited from the audience. To open the session, A. Ulses (IAEA) expressed satisfaction with the organisation and smooth-running of the Workshop, and complimented the staff at KAERI on their efforts in this regard. The level of attendance confirmed the international level of interest in the theme and objectives of the Workshop, and he pledged continuing IAEA support for the future.

Conclusions and Recommendations

The session topics were wide and various, including advanced reactor modelling, flow mixing issues, boiling and condensation modelling, multi-phase and multi-physics problems, containment analysis, plant application, hydrogen transport and fires, advanced measuring techniques, and single and multiphase flow in rod bundles. Comments arising from the summaries included below.

General

- 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 current format, length and interval between CFD4NRS Workshops were generally considered appropriate, as was the rotation of venues worldwide. Hence no changes were proposed.

- The formula of combining the blind CFD benchmark activity with the occasion of the Workshop was appreciated, giving participants the possibility to display their work (as posters without accompanying papers), discuss their experiences with other participants, and visit the test facility on which the exercise was based. This practice will therefore be continued as far as possible in the future.

- Considerable interest was raised in the proposed forthcoming CFD benchmark on containment modelling and analysis, and to link the activity with CFD4NRS-5, giving people the opportunity to visit the PANDA facility.

- The Panel chairmen, B.L. Smith and D. Bestion, on behalf of the organising committee, promised to pay more attention to the quality of the papers ahead of the CFD4NRS-5 workshop.

Specific

- The nuclear CFD community should be encouraged to apply and further develop Uncertainty Qualification (UQ) methods in regard to their simulations, including uncertainties arising from the numerical solution procedure, the physical models employed, and in the initial and boundary conditions.

- It was noted that CFD is no substitute for properly understanding the basic thermal-hydraulic phenomena involved in the particular numerical analysis being undertaken. The CFD tools should be used instead to quantify the complex interplay between the various physical processes taking place.

- The analytical presentations at the Workshop demonstrated the almost universal application of Best Practice Guidelines in producing CFD simulations, including the use of higher order differencing methods for the fundamental equations. However, in reactor applications, the need for grid sensitivity studies still has to be balanced against the availability of appropriate computational resources.

- A similar code of practice in conducting experiments appears not to be so widespread, but the need for test data to be accompanied by error bars as a guide to measurement uncertainty is still to be encouraged for code validation tests.

Several presentations showed that CFD was being used to guide the design of experiments in several key areas, and in the placement of instrumentation. This is a very welcome development.

CONTENTS OF THE PROCEEDINGS

(Full text in electronic version only)

- CFD Analysis of the OECD/NEA-KAERI Rod Bundle Benchmark Exercise with a Split Vane by RANS Turbulent Models of START-CCM+ 6.06, *H. S. Kang, S. K. Chang, C.-H. Song*

Poster Paper Session 2

- Computations of Transient Natural Circulation on PNL 2 by 2 Test Bundle Experiments, *H. Kwon, S. J. Kim, K. W. Seo, D. H. Hwang*
- Experimental Study on the Thermal Stratification and Natural Circulation Flow inside a Pool, *S. Kim, D. E. Kim, C. H. Song*
- Simulation of Thermal Stripping at T-Junction Pipe Using LES with Mode Parameters and Temperature Diffusion Schemes, *A. Nakamura, Y. Utanohara, K. Miyoshi, N. Kasahara*
- Numerical Analysis of the Passive Condensation Cooling Tank (PCCT) using the CUPID Code, *S. J. Lee, H. K. Cho, K. H. Kang, S. Kim, H. Y. Yoon*

Video Session

- Boiling Behavior of Droplets Impinging on Heated Liquid Metal Surface, *T. Yasui, S. Someya, K. Okamoto*
- Cross-mixing in a Fuel Rod Bundle, Enhanced by Functional Spacer Grids Portraits of Liquid Film Flows, *A. Ylönen, H.-M. Prasser*
- Portraits of Liquid Film Flows, *M. Damsohn, D. Ito, R. Zboray, H.-M. Prasser*
- The Best of Wire-mesh Sensors — Inspirations for their Future Use, *H.-M. Prasser*
- Numerical Modeling of Pool and Flow Boiling, *B. Niceno, Y. Sato*
- ANSYS Fluent results for the split type spacer grid geometry of the OECD/NEA MATiS-H Benchmark, *A.A. Matyushenko, A.V. Garbaruk, S. Jain, T. Frank*

Program Book Technical Papers

Session 1 – Advanced Reactors

- Multi-dimensional temperature distribution in PCCT (Passive Condensation Cooling Tank) and PCHX (Passive Condensation Heat Exchanger) of PAFS (Passive Auxiliary Feedwater System), *Byoung-Uhn Bae, Seok Kim, Yu-Sun Park, Bok-Deuk Kim, Kyoung-Ho Kang*
- Uncertainty Quantification Scheme in V&V of Fluid-Structure Thermal Interaction Code for Thermal Fatigue Issue in a Sodium-cooled Fast Reactor, *Masaaki Tanaka*
- CFD Multi-Physics Analysis of Fuel Bundles under Accidental Conditions for New Fuel Designs, *Yiban Xu, Jin Yan, Kun Yuan, Chun Fu, Peng Xu, Sumit Ray*

Session 2 – Condensation

- Two-Phase CFD PTS Validation in an Extended Range of Thermo Hydraulics Conditions Covered by the COSI Experiment, *Pierre Coste, A. Ortolan*

- Validation of a CFD Model for Steam Condensation in the Presence of Non-condensable Gases, *A. Dehbi, F. Janasz, B. Bell*

- CFD Simulation of Air-Steam Flow with Condensation, *L. Vyskocil, J. Schmid, J. Macek*

- CFD Modelling and Validation of Wall Condensation in the Presence of Non-condensable Gases, *G. Zschaeck, T. Frank and A. D. Burns*

Session 3 – Boiling/Bubbly Flow (1)

- Implementation and Validation of Two-Phase Boiling Flow Models in OpenFOAM, *Kai Fu, Henryk Anglart*

- Development and Validation of a Boiling Model for OpenFOAM Multiphase Solver, *J. Peltola, T.J.H. Pättikangas*

- CFD for Subcooled Flow Boiling: Coupling Wall Boiling and Population Balance Models, *Eckhard Krepper, Roland Rzehak, Conxita Lifante, Thomas Frank*

- Application of New Closure Models for Bubble Coalescence and Breakup to Steam-Water Pipe Flow, *Yixiang Liao, D. Lucas, E. Krepper*

Session 4 – Bundle Flow (1)

- OECD/NEA – MATiS-H Rod Bundle CFD Benchmark Exercise Test, *Seok-Kyu Chang, Seok KIM, Chul-Hwa Song*

- Analysis of the Flow Down and Upwind of Split-Type Mixing Vanes, *Ulrich Bieder*

- The OECD/NEA MATiS-H Benchmark – CFD Analysis of Water Flow through a 5x5 Rod Bundle with Spacer Grids using ANSYS Fluent and ANSYS CFX, *Th. Frank, S. Jain, A.A. Matyushenko, A.V. Garbaruk*

Session 5 – Bundle Flow (2)

- Sensitivity Studies on CFD Analysis for Heat Transfer of Supercritical Water Flowing in Vertical Tubes, *Attila Kiss, Dr. Attila Aszódi*

- Validation of CFD Method in Predicting Steady and Transient Flow Field Generated by PWR Mixing Vane Grid, *Jin Yan, Michael E. Conner, Robert A. Brewster, Zeses E. Karoutas, Elvis E. Dominguez-Ontiveros, Yassin A. Hassan*

- Using the DINUS Code for Direct Numerical Simulation of Hydrodynamic Processes in VVER-440 Fuel Rod Bundles, *Yury.V. Yudov*

Session 6 – Hydrogen Transport and Fire

- CFD Analysis of a Hypothetical H2 Explosion Accident between the HTTR and the H2 Production Facility in JAEA, *Hyung Seok Kang, Sang Baik Kim, Min-Hwan Kim and Hee Cheon No*

- Passive Auto-Catalytic Recombiner Operation Validation of a CFD-approach against OECD-THAI HR2-test, *Stephan Kelm, Wilfried Jahn, Ernst-Arndt Reinecke, Hans-Josef Allelein*

- Development of CFD Based Numerical Tool for Addressing Hydrogen Transport and Mitigation Issues in the Containment of Nuclear Power Plants, *Vikram Shukla, Sivagangakumar P., Sunil Ganju1, Anil Kumar K. R. S. G. Markandeya*

- Validation of Coupled BVM-EDM Combustion Model in ANSYS CFX for Hydrogen Combustion Calculation during Postulated Severe Accidents in NPP, *Sudarat Worapittayaporn, Luciana Rudolph*

Session 7 – Multi-scale & Multi-physics Analysis

- A CMFD-model for Multi-scale Interfacial Structures, Susann Haensch, Dirk Lucas, Eckhard Krepper, Thomas Höhne

- Coupling of CFD Code with System Code and Neutron Kinetics Code, *L. Vyskocil, J. Macek*

- Development of Domain Overlapping STH/CFD Coupling Approach for Analysis of Heavy Liquid Metal Thermal Hydraulics in TALL-3D Experiment, *Marti Jeltsolv, Kaspar Kööp, Pavel Kudinov, Walter Villanueva*

- Code_Saturne Integral Validation on ROCOM Test for Heterogeneous Inherent Boron Dilution Transient, *B. Gaudron, S. Jayaraju, S. Bellet, P. Freydier, D. Alvarez*

Session 8 – Plant Applications (1)

- Large-scale Turbulent Simulations of Grid-to-rod Fretting, J. Bakosi, N. Barnett, M. A. Christon, M. M. Francois, and R. B. Lowrie

- Optimisation of the Atucha-II Fuel Assembly Spacer Grids, *D. Melideo, F. Moretti, F. Terzuoli, F. D'Auria, O. Mazzantini*

- Development of a CFD Model for Investigation of Atucha-II Containment, *D. Melideo, L. Mengali, F. Moretti, W. Giannotti, F. Terzuoli, F. D'Auria, O. Mazzantini*

- CFD Simulations for APR+ Reactor Design, *Seung-Geun Yang, Eung-Jun Park*

Session 9 – Bundle Flow (3)

- Experimental Studies on Sub-cooled Boiling in a 3x3 Rod Bundle, *Frank Barthel, Ronald Franz, Eckhard Krepper, Uwe Hampel*

- Experimental study of a Simplified 3 X 3 Rod Bundle using DPIV, *Elvis Dominguez-Ontiveros, Y Hassan, R. Franz, R. Barthel, U. Hampel*

- 3x3 Rod Bundle Investigations. Part II: CFD Single-Phase Numerical Simulations, *C. Lifante, B. Krull, Th. Frank, R. Franz and U. Hampel*

Session 10 – Plant Applications (2)

- CFD Predictions of Standby Liquid Control System Mixing in Generic BWR, *Christopher Boyd, R. Skarda*

- Numerical Simulation of the Insulation Material Transport to a Pressurised Water Reactor Core under Loss of Coolant Accident Conditions, *Thomas Hoehne, Alexander Grahn and Sören Kliem*

- CFD Analysis of the Temperature Field in Emergency Pump in LOVIISA NPP, *T. Rämä and T. Toppila, T. Kelavirta and P. Martin*

- Numerical Simulation of Two-Phase Critical Flow in a Convergent-divergent Nozzle, *Masahiro Ishigaki, Tadashi Watanabe, and Hideo Nakamura*

Session 11 – Boiling/Bubbly Flow (2)

- Visualisation of High Heat Flux Boiling and CHF Phenomena in a Horizontal Pool of Saturated Water, *In-Cheol Chu, Hee Cheon NO, Chul-Hwa Song*

- Experimental Data on Vertical Air-Water Pipe Flow Obtained by Ultrafast Electron Beam X-Ray Tomography Measurements, *Dirk Lucas, M. Banowski, D. Hoppe, M. Beyer, L. Szalinski, F. Barthel, U. Hampel*

- On the Effects of Orientation Angle, Subcooling, Mass Flux, Heat Flux, and Pressure on Bubble Departure Diameter in Subcooled Flow Boiling, *Rosemary Sugrue, T. McKrell, J. Buongiorno*

- Validation of Models for Bubbly Flows and Cap Flows using One-Group and Two-Group Average Bubble Number Density, *G.H. Yeoh, S.C.P. Cheung, J.Y. Tu, D. Lucas, E. Krepper*

Session 12 – Mixing

- Addressing the Accuracy Quantification issue for CFD Investigation of In-Vessel Flows, *F. Moretti, Francesco D'Auria*

- Investigation of the Thermal Mixing in a T-Junction Flow with Different SRS Approaches, *Mikhail Gritskevich, A. V. Garbaruk, and F. R. Menter*

- Large-eddy Simulations of Stratified Flows in Pipe Configurations Influenced by a Weld Seam, *David Kloeren, Mario Kuschewski, Eckart Laurien*

- CFD Analysis on Localised Mass Transfer Enhancement in the Downstream of an Orifice, *Jinbiao Xiong, Xiangfeng Pan, Seiichi Koshizuka, Lefu Zhang, Xu Cheng*

TECHNICAL SESSION SUMMARIES

Session 1: Advanced Reactors (Session Co-Chairs: D. Bestion and B.D. Chung)

The session on "Advanced reactors" had three papers on very different topics, on an experiment, on uncertainty of CFD, and on multi-scale simulations.

The first paper by Bae et al. presented an experiment about a pool heat exchanger (HX). Measurements are made in the tube of the HX, in the pool, and the heat flux through the tube wall was also measured. Some estimation of the experimental uncertainty was given. The "Pool HX" was identified by the Writing Group 3 of OECD-WGAMA on "Extension of CFD to two-phase reactor safety issues" as an issue where two-phase CFD can bring a real benefit by predicting the mixing in the pool in presence of temperature stratification. The temperature field in the pool was measured and the void fraction will be measured but no velocity measurement is planned. Although the experiment provides good quality and original data which may be used for CFD validation, it was mentioned that a more precise validation would require also velocity measurements.

The second paper by Tanaka investigated the Uncertainty Quantification (UQ) of the CFD simulation of thermal fatigue. UQ is a very important issue for the application of CFD to Safety and this is one of the first pioneering analyses of this kind and the first paper on uncertainty presented at CFD4NRS workshops. The nuclear CFD community is encouraged to follow this example and to apply and further develop the UQ methods. Here the work focused on uncertainty due to the numerics by looking at the GCI (grid convergence Index) and it should be extended to uncertainty due to physical models, to initial and boundary conditions, and so on…Further work of this kind and many papers on UQ are expected at the next CFD4NRS Workshop in 2014.

The third paper by Xu et al. is a multi-scale analysis of Departure from Nucleate Boiling (DNB) including a sub-channel code and a CFD code. DNB is a safety issue but it is also a design issue and this work is a CFD application to design investigations. The requirements for application of BPG and for Assessment may be less strict for application to design compared to application to safety and actually no BPG and no assessment was mentioned. CFD is here supposed to provide at least good qualitative trends when trying to improve the design of fuel assemblies. The fact that industry (here Westinghouse) finds some benefit in applying CFD indicates that the maturity of this new technology is not so bad. This was confirmed by the 5th Invited Lecture given by Ikeda who concludes that CFD can play a significant role in designing high efficiency PWR spacer grids. Two-phase CFD is not yet very accurate and is not more accurate than 1D models or sub-channel models but it can see phenomena at a smaller scale and this is already a benefit. In this Workshop, for the first time in the CFD4NRS series, a limited number of papers were specifically devoted to the use of CFD in reactor design. These provide an interesting perspective on the maturity of the technology.

Session 2: Condensation (Session Co-Chairs: P. Coste and Y. Ishiwatari)

This session included one paper dealing with NEPTUNE_CFD and three papers dealing with ANSYS commercial codes. With NEPTUNE_CFD, it was about Pressurised Thermal Shock. With ANSYS codes, the three papers were about wall condensation in the presence of non-condensable gases.

The first presentation by dealt with a two-phase CFD validation in the context of the Pressurised Thermal Shock. The validation is based on an extended range of thermal hydraulics conditions covered by the COSI experiment. The experiment represents a cold leg scaled from a PWR, with its emergency core cooling systems under LOCA conditions. The results obtained by NEPTUNE_CFD on fifty different runs were compared to the experiment, generally in a first step. Then six very different configurations were analysed in more detail, discussing the reliability of the simulations regarding local parameter (temperatures) and illustrating various CFD potentialities but also some weak points.

The second presentation dealt with the integration in FLUENT of a user-defined model for wall condensation in the presence of non-condensable gas. The model is derived from first principles as a sink term for the mass, momentum, species and energy conservation equations. The validation was conducted step by step: first laminar and turbulent forced flows along a cold plate. Then more challenging simulations of the condensation along a vertical cylinder tube standing in a closed vessel were compared to analytical and experimental correlations. The simulations improvement obtained with the introduction of a “suction term” was discussed. As in the two next papers, the model assumes that the thermal resistance of the liquid film is negligible. This point has been discussed by the audience but it was justified by the presenters who recalled that the predictions are only valid for large mass fractions of the non-condensable (above 20%).

The third presentation dealt also with the integration in FLUENT of a user-defined model. This condensation model consists of two parts: condensation in volume and condensation on the wall. Condensation in volume is done by “return to saturation in constant time scale”. Condensation on the wall is calculated from diffusion of steam through a layer of non-condensable gases near the wall. The model is then successfully compared to the CONAN and PANDA experimental data.

The fourth presentation dealt with the description and validation of a model implemented in CFX. This model is basically similar to the previous model for wall condensation. It employs a mass sink at isothermal walls or conjugate heat transfer domain interface where condensation takes place. The model is validated using the data reported by Ambrosini et al. (2008) and Kuhn et al. (1997).

Session 3 Boiling/Bubbly Flow (1) (Session Co-Chairs: D. Lucas & G.H. Yeoh)

This session included four papers discussing the implementation, application and validation of CFD models to boiling (3 papers) and condensing (1 paper) flows.

The first paper dealt with the implementation of new models for the wall heat partitioning as well as for the phase change terms in nucleate boiling in the open source environment OpenFOAM. A transport equation for the interfacial area concentration was also implemented to predict the bubble size in the bulk liquid flow. Validation of the model was performed against available data such as from the DEBORA experiment. With regards to the wall heat partitioning model, the three important parameters of bubble departure, bubble frequency and active nucleation site density were assessed. The force balance model was found to yield reasonable agreement between predictions and measurements, despite the profile of the Sauter mean diameter. For this reason the question was raised whether the interfacial area concentration is a well suited transport equation for this case. More work is still required to determine suitable models for the bubble frequency and active nucleation site density. The need of experimental data set which includes the complete information on the boiling process was stressed out.

The second paper also focused on the development of models for the wall heat partitioning in the open source environment OpenFOAM. Thermodynamic non-equilibrium and compressibility of gas and liquid phases were accounted by adding enthalpy equations to solve for the respective phases. Validation of the model was first performed on dispersed bubbly air flow with the consideration of different interfacial forces against data from the DEDALE experiment. The boiling model was subsequently validated against data from DEBORA experiment. Computed results were also compared against those obtained from ANSYS-Fluent.

The third paper assessed the models for wall heat partitioning developed in ANSYS-CFX. These models are coupled with a population balance based on the inhomogeneous MUSIG (MUltiple SIze Group) model to predict the bubble size distribution in the bulk liquid flow. Separation of large and small groups of bubbles was considered, and consideration of different gas velocities apportioned for these two groups of bubbles. Validation of the results was performed against data from a DEBORA experiment. Local and axial predictions through these coupled models demonstrated the capability of observing the increase of the bubble size after leaving the heated wall as well as the change of gas volume fraction profiles transiting from a wall to a core peak with increasing inlet temperature.

The fourth paper presented the validation of new closure models for bubble coalescence and break-up which were previously implemented into the ANSYS-CFX code, for condensing steam-water pipe flows. The inhomogeneous MUSIG (MUltiple SIze Group) model, considering the separation of large and small bubble into two velocity groups, was adopted to predict the bubble size distribution in the bulk liquid flow. Validation of the results was performed against experiment performed at Helmholtz- Zentrum Dresden-Rossendorf. Various heat transfer correlations were tested to assess the inter-phase heat transfer models for the condensation of bubbles. Sensitivity studies were also performed for the inlet liquid temperatures. The new closure models were shown to perform better than the default models in ANSYS-CFX. Nevertheless, both models over-estimated the break-up rate as the gas bubbles travelled downstream.

All papers showed a clear progress of CFD capabilities for the simulation of wall boiling and bubbly flows with phase transfer. Reasonable agreement was observed for some cases, but up to now no general approach is available, which is applicable over a wide range of flow parameters. Hypothesis for the reasons of observed deviations were given, but due to the complex interaction between different phenomena they cannot be directly assigned to weaknesses in the modelling of a special phenomenon. For further improvements of the models new experimental data are required containing complete information on the local flow characteristics.

Session 4: Bundle Flow (1) (Session Co-Chairs: C. Boyd and E. Merzari)

Session 4 contained three papers that were focused on Fuel Bundle modelling and specifically the MATHIS-H benchmark exercise. All the presentations were detailed, well thought out, and represented significant amounts of work.

The first paper provided details about the MATIS-H experiment, with additional information provided about how the experiment was conducted and how some of the assumptions were verified. Questions related to the measurement locations and the outlet plenum were addressed. In particular a valuable series of tests in which the location of the spacer grid was fixed and the position of the measurement plane altered were presented proving a negligible influence of the outlet on the measurements. This presentation provided additional insights into the test procedures and rationale and is very beneficial as a review for those interested in the benchmark study.

The second paper looked in detail at the flows associated with the benchmark study and provided insights into modelling techniques that are needed to address flow effects both upstream and downstream of the mixing grids as well as turbulence modelling issues. Issues such as upstream flow and turbulence, fabrication of the mixing vanes, and the downstream boundary location were studied by comparing experimental data with both a series of LES simulations (performed using the CEA code Trio U) and RSM results obtained at EDF. Studies of bare tube bundles are used to represent the flow upwind of the mixing grid. The focus of the paper and the presentation is on flow physics: turbulent mixing is characterised by an analysis of the Reynolds stress tensor. The authors emphasise the presence of anisotropy and its influence on secondary flow structures. The region downstream of the grid spacers is studied by directly comparing the predictions to the MATIS-H data. While the LES results showed reasonable agreement with the mean velocity the author pointed out how the rms values were underestimated due to the stabilisation techniques employed in the simulation. The results are used to evaluate the need to include the outlet plenum as part of the model. Overall the paper offered valuable insight into the challenges of a LES simulation of the flow past a grid spacer with a knowledgeable focus on flow physics.

The final paper included a comprehensive study using ANSYS FLUENT and ANSYS CFX. Details of the overall modelling process were included to provide insights into the procedure used to solve the problem in a systematic way. Studies included upstream flows, an analysis of a small section for assessment and development of best practices, and finally the full rod bundle modelling. Two separate codes were used with the same mesh and general turbulence options for a very informative code-to- code comparison. Moreover several advanced turbulence modelling options have been tested, including SAS-SST and SAS-Zonal LES. Of notable interest was also the comparison of LES bare bundle results with experimental inlet data and other turbulence models results. Computational results are compared to experimental data. Overall the study presented an impressive amount of work and it provides significant insight and lessons learned from the benchmark.

Overall, the Session provided a good overview of the MATHIS-H experiments and two significant efforts to predict the downstream flow behaviour. This session provides some key reference material that should be consulted by those considering the MATIS-H benchmark study.

Session 5: Bundle Flow (Session Co-Chairs: K. Ikeda and J. Yan)

There were three papers in this session. The papers focus on validating CFD codes for bundle flow. The first paper dealt with the effect of the specified boundary conditions on the CFD results. It consists of a number of very basic simulations. The validation against the test data is needed for those parameters.

The second paper contrasted the steady and transient CFD approach to modelling the flow in a PWR 5x5 fuel assembly. The paper described the fundamental link between Grid-To-Rod-Fretting (GTRF) and the characteristics of the transient flow field. For the first time, an attempt was made to validate the transient CFD predictions against test data in an actual PWR fuel assembly. The paper also outlined the key parameters on which to concentrate when comparing the calculated and measured transient flow field. The paper demonstrated an effective method of evaluating GTRF in fuel assembly design.

The third paper dealt with the validation of the CFD code DINUS against the measurement data obtained from experiments on the full-scale hydraulic model of the VVER-440 fuel rod bundle test at Reynolds number 50’000. Good agreement between computational and experimental data was demonstrated for the velocity and turbulence intensity in the axial direction at different downstream distances from the spacer grid.

Session 6: Hydrogen Transport and Fire (Session Co-Chairs: C. Boyd and Y.H. Ryu)

Session 6 included four papers. Three of these focused on containment modelling issues, and one paper considered a hypothetical hydrogen explosion and its impact on neighbouring buildings. The presentations were well prepared and represented significant amounts of work.

The first paper considered the impact of a hydrogen explosion accident on a neighbouring facility and the ability of CFD to make these types of predictions. The applicability of the CFD method was qualitatively confirmed by comparison with some limited test data. Several configurations were considered and basic best practice guidance was employed. Only simplified geometries were considered at this time, but the method looked promising for application to a future production facility.

The second paper focused on modelling issues associated passive auto-catalytic recombiners (PARs), and included validation of the CFD approach against the ThAI HR2 test. A modelling strategy for PARs was outlined which involved combining a detailed PAR model linked to the CFD code ANSYS CFX. The model was benchmarked against the ThAI test data and good qualitative agreement was found. Some limitations and future work were discussed.

The third paper also focused on modelling issues associated with PARs. The CFD tool FLUIDYN-MP was modified with user-defined code to address bulk condensation and evaporation, wall condensation, and the chemical reaction associated with PAR operation. Validation was carried out using some available data from the PANDA test facility and the results were in general agreement. Several parametric studies were also included.

The fourth paper focused on the issue of hydrogen combustion. In this paper, two hydrogen combustion models (the Burning Velocity Model and the Eddy Dissipation Model) available in the ANSYS-CFX 12.1 code were combined and the approach validated against two sets of experiments. In addition, some code-to-code comparisons were made and a set of sensitivity studies completed. Some additional work was also identified. This paper represents a significant amount of work.

Session 6 provided a sampling of hydrogen modelling issues and was especially relevant for those interested in hydrogen mitigation issues during severe accident scenarios. Although sensitivity studies are generally undertaken, and codes benchmarked against test data, there is still room for improvement in the documentation of the application of best practice guidance to ensure that the codes are used appropriately.

Session 7: Multi-scale & Multi-physics Analysis (Session Co-Chairs: U. Bieder and H. Nakamura)

Three topics were addressed in this session: (i) the modelling of the interfaces in multi-scale bubbles (1 paper); (ii) the coupling of CFD codes to system codes (2 papers); and (iii) the treatment of inherent homogeneous boron dilution (1 paper). The first paper recognised that, for many multiphase flows, segregated and dispersed flow structures are encountered simultaneously. Transitions between such morphologies, characterised by the different scales of the interface structures, play an important role. The paper introduced a new CMFD strategy for such generalised two-phase flows: GENTOP. The inhomogeneous Multiple Size Group (MUSIG) model was extended by adding a continuous gas phase, which incorporated all the gas structures large enough to be resolved within the computational mesh. A free surface detection model and a generalised formulation for the interfacial transfer models were introduced for this purpose. During discussion, it was stressed that the turbulence model for the continuous phase must be consistent with the modelling hypotheses employed for the disperse phase.

Both presentations on the coupling of CFD codes to system codes showed the importance of correctly interpreting the detailed 3-D information derived from CFD calculations to the system code simulation. Typical areas where 3-D effects might influence the system code results are the lower plenum, e.g. in a MSLB accident in PWRs, or in pool-like components of heavy-liquid-metal (HLM) reactors. However, there is still a lack of experimental data for validation. For PWRs, some experimental data are available, whereas for HLMs an new study is underway at KTH, Stockholm. In the second paper, a coupling interface between the CFD code Fluent and the system code Athlet, internally coupled with the neutron kinetics code Dyn3D, was described. Due to the strong coupling between thermal hydraulics and the neutron kinetics, scenarios such as those deriving from MSLB events cannot be represented in successive calculation steps. Instead, an explicit coupling of overlapped computational domains is used in this work. As demonstration, both coupled and uncoupled approaches were studied of a test carried out at the Temelin NPP (VVER-1000).

The third paper described a pre-test analysis performed to provide insights on 3-D flow in the small cylindrical-shaped test section of the TALL-3D test facility. The test section is composed of two parallel equal-diameter, equal-height pipes, each equipped with a heater, flow meter and heat exchanger. Inlet and exit flow conditions for the 3D-pool of the facility were first estimated using the 1-D RELAP5 code, and were then passed to the Star-CCM+ code to estimate the 3-D flow in the pool tank. Some flow fluctuations appeared between the three test section pipes, with oscillatory flow within the 3D pool where the two upward streams meet -- the high-temperature fluid going up along heated side walls and low-temperature fluid entering upwards from the bottom along the centreline. In-depth analysis is foreseen once construction of the TALL-3D test facility is complete, and detailed temperature distributions measured in the 3D-pool.

The final paper describes the EDF-developed CFD code Saturne and its validation against the 3-D coolant mixing experiment performed at the ROCOM test facility at HZDR, Germany. Saturne is an open-source CFD code. The ROCOM facility simulates multi-dimensional coolant flows within a PWR pressure vessel, especially in the cold legs (with low-temperature coolant injection), the downcomer, and the lower plenum and core, with a focus on the boron dilution process. Tests of interest were selected based on a PIRT approach to rank the major phenomena, such as coolant mixing in the cold legs and downcomer, and the coolant distribution in the lower plenum. Saturne was able to reproduce the 3-D flow conditions observed in the ROCOM tests well, provided the OECD Best Practice Guidelines were employed, these including sensitivity studies on mesh distribution and the numerical schemes used. Though a URANS model was used in this work, a recommendation was received from the audience to test other turbulence models in order to improve the accuracy of the numerical results.

Session 8: Plant Applications (1) (Session Co-chairs: Y. Hassan and K.D. Kim)

In this session four papers are presented. The following addresses the presentations:

The first presentation entitled Large-Scale Turbulent Simulations of Grid-to-Rod Fretting demonstrated the application of CFD to predict the sigle phase flow in 5x5 fuel bundle. Implicit large-eddy simulation, detached-eddy simulation, and unsteady Reynolds-averaged Navier-Stokes calculations with a Spalart-Allmaras model are applied to the problem. However, a need for finer nodalisation is indicated to achieve a reasonable prediction with the data.

The second presentation is entitled optimisation of the Atucha-II fuel assembly spacer grids. This work deals with a demonstrative application of Computational Fluid Dynamics (CFD) for design optimisation .A commercial CFD code was applied to simulate the flow within Atucha-II reactor coolant channels and to assess the resulting pressure losses. Two types of grid were simulated, i.e. the so-called "elastic spacer" and "rigid spacer", which are made from different materials and have totally different geometries. Both designs are planned to be adopted in Atucha-II reactor. The turbulence model and the grid need to be addressed to have a confidence in the results. No experimental validation is discussed.

The third presentation is entitled development of a CFD model for investigation of Atucha-II containment. The study deals a demonstrative application of Computational Fluid Dynamics (CFD) for Containment analyses. A commercial CFD code was applied to simulate the flow distribution within the Atucha-II containment and to test the wall condensation model. The CFD results have been compared with the TH-SYS code results. A 3-D model in CFX code without enough cells for full containment is implemented. A condensation model is used without any validation.

The fourth presentation is entitled CFD for APR+ reactor design. FLUENT code is used simulate the flow field in APR+ core. The computational domain is limited to from Low End Fitting (LEF) to Upper End Fitting (UEF) of APR+ core. The 257 advanced fuel assemblies are considered in the model. The intention is to predict the hottest location in APR+ core. Qualification of the code to predict the complex flow behavior with a limited number of cells is not addressed. During the presentation, questions are pointed to the deficiency of the fidelity of the predictions without the intensive further assessments.

Session 9: Bundle Flow (Session Co-Chairs: K. Okamoto and B. Smith)

This session consisted of three papers: all were interrelated, and based on boiling experiments performed, or to be performed, at HZDR, Germany in a 3x3 rod bundle geometry.

The first paper served as an introduction to the work programme, the measuring techniques, and the preparations being made in advance in both the experimental and analytical contexts by Texas A&M University and ANSYS Germany. Highlights from the HZDR experiments using the refrigerant RC318 with the use of high-speed (2000 fps) X-ray tomography, and a gamma-ray densitometer to measure void fractions. Attention was paid to the spatial resolution and uncertainties of the measurements, and to the effects of by-pass flow in the circumferential region.

The experimental techniques used at Texas A&M were described in detail in the second paper. A unique feature of the tests is the use of p-cymene as the (single-phase) coolant and pyrex glass for the rods. These materials have closely matching refractive indices, so the light sheet from the LDA equipment can measure velocities in the fluid even if a rod obstructs the view. CFD-grade data were produced from the tests.

Very detailed pre-test simulations, carried out using ANSYS CFX, were reported in the third paper. Best Practice Guidelines were followed. The calculations revealed that the rod support grid, which was specifically designed to minimise the blockage effect to the flow, nonetheless had a non-trivial influence on the flow field. Also, it was noted that there was an effect due to the geometry of the inlet plenum upstream of the rods, which necessitated separate modelling to provide the inlet boundary conditions for the rod bundle simulations.

The attempt to validate a CFD simulation code using precise experimental data is a very important component of nuclear reactor safety evaluation, and similar projects should be carried out in other countries. The particular collaboration between the three organisations represented in this session of the workshop is a good example of the contact necessary in advance for defining an experimental programme for the validation of CFD codes, and acts as a role model for communication between experimental and CFD specialists, a theme central to the CFD4NRS workshop series.

Session 10: Plant Applications (2) (Session Co-Chairs: R. Schultz and J.J. Jeong)

In this session, four papers were presented on CFD application to plant design and operation.

In the first paper a set of CFD predictions were undertaken for a full-scale BWR/4 lower plenum to show the influence of the overall system flow rate on the stratification and mixing behaviour of the SLCS flows during an ATWS event. A range of reactor flow conditions were studied to map out the stratification and entrainment behaviour for the injected solution from the SLCS as a function of the flow rates through the reactor. Modelling choices were outlined and discussed in the light of best practice guidance principles. Assumptions and limitations in the current approach were discussed to ensure the results are put into the proper perspective. The predictions shed additional light on previous test programs, and the lessons learned include the potential significance of the geometric and flow field details in the lower plenum. Overall, the CFD predictions add significant insights into the SLCS behaviour in the lower plenum of a BWR, and these predictions will provide system code modellers additional information to support the modelling of SLCS flows during ATWS scenarios.

In the second paper, a 3-D, time-dependent, multi-phase flow problem was studied applying the CFD code ANSYS CFX. Best practice guidelines were employed to identify the appropriate grid size, turbulence model and multiphase model to use. The spacer grids were modelled as a strainer, completely retaining all the insulation material reaching the uppermost spacer level. In this model, which was validated against experiments at HTWS Zittau, the accumulation of the insulation material was seen to give rise to the formation of a compressible fibrous cake, the permeability of which to the coolant flow was calculated in terms of the local amount of deposited material and the local value of the superficial liquid velocity. The CFD simulations have shown that after starting the sump mode, the ECC water injected through the hot legs flows down into the core at so-called “breakthrough channels” at the outer regions of the core, a phenomenon first recognised in the UPTF experiments.

In the third paper, a CFD model has been developed for one of the pump rooms of the Loviisa NPP, whose geometry was scanned with a laser, the data from which were converted to detailed provide a detailed 3D geometry. A computation grid was created using the information, and the flow and temperature field in the enclosure were calculated. Grid sensitivity studies were made, and the model was then validated against data from the power plant tests. After the validation phase, postulated accident conditions were simulated numerically, and included sensitivity analyses concerning the operation of the cooling fan units.

In the fourth paper, flow from a jet nozzle was analysed numerically using the CFD code FLUENT, quantitatively assessing the performance of the code predictions against measured data in the form of mass flow rates, pressure and void fractions. A cavitation model is applied in additions to evaporation and condensation models. Estimations of mass flow rate agree with the experimental results in the case of low axial pressure difference. On the other hand, the mass flow rates are under-estimated when the axial pressure difference is high. It was found that predictions of the mass flow rate are much improved by taking into account the effect of the wall vapour generation.

Session 11: Boiling/Bundle Flow (Session Co-Chairs: H.-M. Prasser and Tadashi Morii)

There were four papers in the session, all of high quality. There were important questions posed by the audience for each of these presentations.

In the first paper, novel experimental results were presented on the observation of the formation of dry areas at a heated surface leading to DNB. This work significantly contributes to the understanding of CHF. The technique used was to use total light reflection as a criterion for the presence of surface wetting. It was found that nucleation at the periphery of a dry spot was the main mechanism leading to a persistent dryout condition. Questions concerning resolution, and the fact that adsorption layers cannot be resolved by optical means, were convincingly answered by stating that adsorption layers are not making a contribution to heat transfer.

The second paper reported first results on flow structure investigations of a gas-liquid upwards flow in a 50 mm pipe obtained by ultra-fast X-ray tomography. This novel instrumentation provides rich primary information (comparable to wire-mesh sensors) but in a non-intrusive way, which makes it a very powerful technique for many two-phase flow studies, Main focus of current and future work is directed towards the improvement of the signal processing as well as application to high- pressure/high-temperature and non-adiabatic flows. Ultra-fast X-ray tomography is likely to become one of the leading techniques for the validation of multiphase CFD.

In the third paper, an experimental study on the departure diameter of bubbles in sub-cooled boiling was presented. Existing models were reviewed, and deviations, especially in the case of a downwards- oriented heated wall, were found to be excessive. The work is relevant to the refinement of heat flux splitting models, such as that of Kurul & Podowski, for the estimation of boiling heat transfer, a model widely used in two-fluid boiling processes.

The fourth paper dealt with the development of a three-fluid model for gas-liquid flows, splitting the gaseous phase into two bubble size groups, was described. Selected test runs of the MTLoop and the TOPFLOW DN200 vertical test section were successfully analysed using this model. Main phenomena observed in the tests were reproduced qualitatively and in acceptable quantitative agreement. Common aspects of the two-group interfacial area density transport model of Ishii were brought up in the discussion following the presentation.

All three experimental papers presented novel results relevant for the advance of CFD for nuclear applications especially important for CFD code validation. The most significant progress was represented by the success in providing a new type of CFD grade instrumentation by the commissioning of the fast X-ray tomograph in HZDR. The optical technique employed to detect dry patches closes an important gap in the observation of the fundamental processes leading to the boiling crisis.

Session 12 – Mixing (Session Co-chairs: T. Höhne and J.C. Jo)

This session consisted of four presentations, each on flow mixing analyses. All papers fully complied with the scope of the workshop, and were relevant to nuclear reactor safety. Two presentations were on thermal mixing in a T-junction, one was devoted to accuracy quantification, and one on the localised mass transfer enhancement.

The first presentation was devoted to accuracy quantification in the CFD investigation of in-vessel flows. An example of the multi-dimensional information that needs to be captured is represented by the time and space distribution of coolant properties (e.g. temperature, boron concentration) at the reactor core inlet, a topic relevant to reactor safety as local perturbations, because of the potential to induce power excursions. An attempt was made to tackle the accuracy quantification issue for this class of data in a somewhat empirical way by proposing a set of parameters that can be used, after a qualitative analysis of measured data and code simulation results, to characterise the "perturbation" of the target variable time and space distributions, and to quantify the deviation of the simulation from the experimental data. The advantage of using many parameters relies on the ability to cover all relevant features of the chosen target variables, and thus provide a more complete assessment and regarding overall agreement.

In the second presentation, an investigation of thermal mixing in a T-junction with different turbulence modelling assumptions was described. The results showed that all models are able to accurately predict the mean and RMS velocity profiles if used in combination with a low dissipation advection scheme. However, when a slightly more dissipative scheme is used, the SAS model yields less accurate results, indicating that this flow does not produce a strong enough flow instability to allow the safe application of the model. In contrast, the DDES and the ELES models show less sensitivity to the numerical scheme adopted. Of prime importance in thermal fatigue analyses is accurate prediction of heat transfer to the walls in the mixing zone. In this respect, the ELES method was shown to produce best agreement with experimental data.

The third presentation focused on Large-eddy Simulations (LES) of stratified flow in pipe configurations influenced by a weld seam. Cracks in the piping material of cooling circuits due to high-cycle thermal fatigue are typically found in the vicinity of weld seams. The speaker showed the influence of a weld seam on the flow fields with regard to fluid-structure-interaction. LES results of T- junction mixing flows and stratified T-junction mixing flows were compared with experimental data obtained using Particle Imaging Velocimetry and Near-Wall LED-Induced Fluorescence.

The last presentation was devoted to a CFD Analysis on localised mass transfer enhancement downstream of an orifice. Mass transfer enhancement is an indispensible element for the occurrence of flow-acceleration corrosion. In this presentation, the classic k-ε and k-ω SST models were used together with wall functions to investigate mass transfer downstream of an orifice in pipe flow. The study revealed that in this application the k-ω SST model was superior to the classic k-ε model.

Because the mass transfer prediction is sensitive to the wall mesh size, a wall unit of y_p30 is recommended. The k-ω SST model was also used to investigate mass transfer in a flow with an elbow installed upstream of the orifice. It was shown that further enhancement of the mass transfer was occasioned by the presence of an upstream elbow.

In this session, it was noted that there was still limited use of BPGs in the applications. Nevertheless, important information was provided concerning the scientific value of understanding mixing processes by means of advanced experiments and CFD simulation was emphasised.