Nuclear Safety
NEA/CSNI/R(2020)5
December 2021
www.oecd-nea.org

Workshop on Experimental Validation and Application of CFD and CMFD Codes to Nuclear Reactor Safety Issues, CFD4NRS-7

Shanghai Jiao Tong University
4-6 September 2018

OECD
BETTER POLICIES FOR BETTER LIVES

NEA
NUCLEAR ENERGY AGENCY

Nuclear Energy Agency

NEA/CSNI/R(2020)5

Unclassified **English text only**

20 December 2021

NUCLEAR ENERGY AGENCY
COMMITTEE ON THE SAFETY OF NUCLEAR INSTALLATIONS

Workshop on Experimental Validation and Application of CFD and CMFD Codes to Nuclear Reactor Safety Issues, CFD4NRS-7

Shanghai Jiao Tong University, 4-6 September 2018

Please note that this document is available in PDF format only.

JT03487712

COMMITTEE ON THE SAFETY OF NUCLEAR INSTALLATIONS

The Committee on the Safety of Nuclear Installations (CSNI) is responsible for the Nuclear Energy Agency (NEA) programmes and activities that support maintaining and advancing the scientific and technical knowledge base of the safety of nuclear installations.

The Committee is a forum for the exchange of technical information and for collaboration between organisations, which can contribute to its activities from their respective backgrounds in research, development and engineering. It has 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 reviews the state of knowledge on important topics of nuclear safety science and techniques and of safety assessments, and ensures that operating experience is appropriately accounted for in its activities. It initiates and conducts programmes identified by these reviews and assessments in order to confirm safety, overcome discrepancies, develop improvements and reach consensus on technical issues of common interest. It promotes the co-ordination of work in different member countries that serves to maintain and enhance competence in nuclear safety matters, including the establishment of joint undertakings (e.g. joint research and data projects), and assists in the feedback of the results to participating organisations. The Committee ensures that valuable end-products of the technical reviews and analyses are provided to members in a timely manner, and made publicly available when appropriate, to support broader nuclear safety.

The Committee focuses primarily on the safety aspects of existing power reactors, other nuclear installations and new power reactors; it also considers the safety implications of scientific and technical developments of future reactor technologies and designs. Further, the scope for the Committee includes human and organisational research activities and technical developments that affect nuclear safety.

Acknowledgements

The Nuclear Energy Agency (NEA) wishes to express its sincere gratitude to the organising committee, the scientific committee and the local organising committee.

Organising Committee

Jinbiao Xiong, Shanghai Jiao Tong University, China, General Chair

Dominique Bestion, Commissariat à l'Energie Atomique, France, Co-Chair

Yassin Hassan, Texas A&M University, United States, Co-Chair

Ghani Zigh, United States Nuclear Regulatory Commission, United States, Co-Chair

Xiang Chai, Shanghai Jiao Tong University, China

Nils Sandberg, Secretariat, Nuclear Energy Agency

Matthias Krause, Observer, International Atomic Energy Agency

Scientific Committee

Henryk Anglart, Royal Institute of Technology (KTH), Sweden

Arnoldo Badillo, Imperial College, United Kingdom

Emilio Baglietto, Massachusetts Institute of Technology (MIT), United States

Dominique Bestion, Commissariat à l'énergie atomique et aux énergies alternatives (CEA), France

Ulrich Bieder, CEA, France

Chris Boyd, United States Nuclear Regulatory Commission (US NRC), United States

Robert Brewster, Westinghouse, United States

Xu Cheng, Karlsruhe Institute of Technology (KIT), Germany

Abdelouahab Dehbi, Paul Scherrer Institute (PSI), Switzerland

Yassin Hassan, Texas A&M, United States

Thomas Hoehne, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany

Yanping Huang, Nuclear Power Institute of China (NPIC), China

Wang-Kee In, Korea Atomic Energy Research Institute (KAERI), Korea

Seiichi Koshizuka, University of Tokyo, Japan

Eckart Laurien, Stuttgart, Germany

Dirk Lucas, HZDR, Germany

Elia Merzari, Argonne National Laboratory (ANL), United States

Fabio Moretti, Nuclear and Industrial Engineering (NINE), Italy

Hideo Nakamura, Japan Atomic Energy Agency (JAEA), Japan

Horst-Michael Prasser, Eidgenössische Technische Hochschule Zürich (ETH Zürich), Switzerland

Afaque Shams, NRG (nuclear research and consultancy group), Netherlands

Jan-Patrice Simoneau, Electricité de France, France

Chul-Hwa Song, KAERI, Korea

Wenxi Tian, Xi'an Jiaotong University (XJTU), China

Jinbiao Xiong, Shanghai Jiao Tong University (SJTU), China

Hongxing Yu, Nuclear Power Institute of China (NPIC), China

Donghui Zhang, China Institute of Atomic Energy (CIAE), China

Ghani Zigh, US NRC, United States

Local Organising Committee (Shanghai Jiao Tong University, China)

Xiaojing Liu

Wenhai Qu

Xiang Chai

Tengfei Zhang

Table of contents

Executive summary ... 7

List of abbreviations and acronyms ... 10

1. Background ... 13

1.1. Scope ... 13
1.2. Organisational aspects ... 13

2. Technical session summaries from the Co-Chairs ... 15

2.1. Session 1: Two-Phase Flow (I) ... 15
2.2. Session 2: Containment (I) ... 16
2.3. Session 3: Experiment technique ... 17
2.4. Session 4: Flow-Induced vibration ... 18
2.5. Session 5: Bundle flow ... 19
2.6. Session 6: Mixing (I) ... 19
2.7. Session 7: Liquid metal ... 20
2.8. Session 8: Two-Phase flows II ... 21
2.9. Session 9: Containment (II) ... 21
2.10. Session 10: Mixing (II) ... 22
2.11. Session 11: Pressurised thermal shock ... 23
2.12. Session 12: Plant application (I) ... 24
2.13. Session 13: UQ and coupling ... 24
2.14. Session 14: Plant applications (II) ... 25

Annex A. Keynote lectures ... 26

Annex B. Poster papers ... 27

Annex C. Technical programme (presented orally) ... 28

Session 1: Two-Phase flow (I) ... 28
Session 2: Containment (I) ... 28
Session 3: Experiment technique ... 28
Session 4: Flow-Induced vibration ... 29
Session 5: Bundle flow ... 29
Session 6: Mixing (I) ... 29
Session 7: Liquid metal ... 30
Session 8: Two-Phase flow (II) ... 30
Session 9: Containment (II) ... 30
Session 10: Mixing (II) ... 31
Session 11: Pressurised thermal shock ... 31
Session 12: Plant application (I) ... 32
Session 13: UQ and coupling ... 32
Session 14: Plant applications (II) ... 32

Executive summary

The Workshop on Experimental Validation and Application of CFD and CMFD Codes to Nuclear Reactor Safety Issues, CFD4NRS-7 was the seventh in a series of workshops focusing on Computational Fluid Dynamics for Nuclear-Reactor Safety (CFD4NRS). It was held in Shanghai Jiao Tong University on 4-6 September 2018 and featured 5 keynote lectures, 45 oral presentations and 6 poster presentations. One hundred and twenty participants attended the workshop and the emphasis was, in a congenial atmosphere, on offering exposure to state-of-the-art (single-phase and multiphase) CFD applications reflecting topical issues in nuclear power plant design and safety, but in particular on promoting the release of high-resolution experimental data to continue the CFD validation process in this application area.

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

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

The third pillar is the evolving use of CFD modelling in multiphase applications, these days known commonly as computational multi-fluid dynamics (CMFD). Here, the challenges are considerable. Not only are the governing equations an order of magnitude more complex than for single-phase applications, but a validation database for which there is genuine three-dimensional involvement remains quite sparse. Of course, multiphase CFD is not used only in the area of nuclear power plant applications, and important developments are taking place in other industrial arenas, such as in the chemical and processing industries, and in environmental studies. Prudence dictates that the CFD4NRS series of workshops should not provide reporting space for such non-nuclear CMFD applications, but should recognise that links with diverse application areas need to be maintained. It is important to ensure that nuclear applications learn from developments in other areas and not repeat mistakes.

Emphasis in this workshop was placed on single-phase and multiphase CFD simulations with a focus on validation in areas such as heat transfer, free-surface flows, direct contact condensation and turbulent mixing. Many papers were related to nuclear power plant-relevant safety issues, such as pressurised thermal shock, critical heat flux, boron dilution, hydrogen distribution in containments, thermal striping and fatigue, and/or advanced design concepts, such as tight-lattice fuel configurations and passive safety options. A few

papers discussed Uncertainty Quantification (UQ) in CFD, which was seen as a promising step, and something that should be further explored.

Although some papers discussed the challenges in obtaining new experimental data, it was felt that this area needs to be emphasised even more, in particular related to two-phase flow, for which sophisticated measurement techniques are required, and for which information is scarce. It is also very important to deepen understanding of the physics before starting a numerical analysis.

Conclusions and recommendations

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

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

Specific recommendations

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

CMFD has been more extensively applied to investigate the two-phase phenomena related to nuclear reactor safety. The closure model development or improvement should be based on CFD-grade experiment data with complete sets of local information. Joint efforts in the community are demanded.

More validation efforts are demanded for CFD simulation on flow-induced vibration. Currently, the vibration frequency can be predicted fairly well. However, the damping of vibration is not so well reproduced.

More emphasis should be given on the experiment measurement error.

General conclusions and recommendations

Delegates were 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 current state of the art and projected future trends. The message received was that “small is good”, and that the workshop should remain strictly focused on CFD issues, and should not broaden its boundaries beyond this area.

The current format, length and interval between CFD4NRS workshops were generally considered still to be appropriate, as was the rotation of venues worldwide. Hence, no changes are proposed in this regard.

The competition of poster presentation should be organised to encourage students to present their work without submitting full papers.

Since previous CFD4NRS workshops had rarely been attended by colleagues from China, great efforts were devoted to let domestic colleagues know about the current workshop.

Over half of the participants were from domestic universities and institutes and, in addition, the workshop attracted participants from many European countries and the United States.

In the panel session at the close of CFD4NRS-7, delegates confirmed their interest in attending the planned follow-up workshop, if possible, and considered the two-year interval between workshops to be appropriate.

It was also recommended that selected papers from the workshop could constitute a special issue of the Nuclear Engineering and Design (NED) journal.

List of abbreviations and acronyms

ADS	Automatic Depressurization System
AIAD	Algebraic interfacial area density
ANL	Argonne National Laboratory (United States)
ASME	American Society of Mechanical Engineers
AULA	Abwaschmodell für unlösliche Aerosole (wash-off model for aerosols)
BIT	Bubble-induced turbulence
BORA	RPV mock-up at 1/5 scale (EDF)
BPGs	Best practice guidelines
BWR	Boiling water reactor
CEA	Commissariat à l'énergie atomique et aux énergies alternatives (French Alternative Energies and Atomic Energy Commission)
CFD	Computational fluid dynamics
CFX	Computational fluid dynamix (commercial software)
CHF	Critical heat flux
CIAE	China Institute of Atomic Energy
CMFD	Computational multi-fluid dynamics
CNRA	Committee on Nuclear Regulatory Activities (NEA)
CSNI	Committee on the Safety of Nuclear Installations (NEA)
DNB	Departure from nucleate boiling
DNS	Direct numerical simulation
EDF	Electricité de France
ETH Zürich	Eidgenössische Technische Hochschule Zürich (Swiss Federal Institute of Technology in Zürich)
FCVS	Filtered Containment Venting System
FIV	Flow-induced vibration
Fluent	Commercial multi-purpose CFD package by ANSYS Inc.
FoM	Figure of merit
FSIFOAM	Finite volume fluid structure interaction solver based on partitioned approach
GEMIX	Mixing experimental facility (PSI)
GENTOP	GENeralized TwO Phase flow
HFT	Hot functional testing

HTC	Heat transfer coefficient
HZDR	Helmholtz-Zentrum Dresden-Rossendorf (Germany)
IAEA	International Atomic Energy Agency
ICC	Ice condenser containment
IRWST	In-Containment Refueling Water Storage Tank
JAEA	Japan Atomic Energy Agency
KAERI	Korea Atomic Energy Research Institute
KIT	Karlsruhe Institute of Technology (Germany)
LBE	Lead-bismuth eutectic
LDV	Laser-Doppler velocimetry
LES	Large eddy simulation
LIF	Laser-induced fluorescence
LIM	Large interface model
LOCA	Loss-of-coolant accident
LOFT	Loss-of-flow transient
LWR	Light-water reactor
MICAS	Mock-up of the hot plenum ASTRID reactor project at CEA
MIT	Massachusetts Institute of Technology (United States)
NEA	Nuclear Energy Agency
NED	Nuclear Engineering and Design (Elsevier journal)
NEPTUNE-CFD	French multiphase CFD code
NINE	Nuclear and Industrial Engineering (Italy)
NPIC	Nuclear Power Institute of China
NRS	Nuclear reactor safety
NSC	Nuclear and Radiation Safety Center (China)
OpenFOAM	Open source Field Operation and Manipulation (open source software for computational fluid dynamics)
PANS	Partially Averaged Navier-Stokes
PAR	Passive Autocatalytic Recombiner
PFM	Pressure Fluctuation Model
PGSFR	Prototype Gen-IV sodium-cooled fast reactor
PIV	Particle image velocimetry
PSI	Paul Scherrer Institute (Switzerland)
PTS	Pressurised thermal shock

PWR	Pressurised water reactor
RANS	Reynolds-averaged Navier-Stokes
RPV	Reactor pressure vessel
SA	Severe accident
SBLOCA	Small break loss-of-coolant accidents
SETCOM	Separate Effect Test for Condensation Modelling
SJTU	Shanghai Jiao Tong University (China)
TAMU	Texas A&M University
THAI	Thermal-hydraulics, Hydrogen, Aerosols and Iodine
TIV	Turbulent Induced Vibration
TUNN	Technical University of Nishny Novgorod (Russia)
UPTF	Upper Plenum Test Facility
UQ	Uncertainty Quantification
URANS	Unsteady Reynolds-averaged Navier-Stokes
US NRC	United States Nuclear Regulatory Commission
V&V	Verification and validation
VOF	Volume-of-fluid
XJTU	Xi'an Jiaotong University (China)

1. Background

This workshop, part of a biennial series of the Nuclear Energy Agency (NEA) and International Atomic Energy Agency (IAEA) sponsored events that began in Garching in 2006, follows the format and objectives of its predecessors in creating a forum whereby numerical analysts and experimentalists can exchange information on the application of computational fluid dynamics (CFD) to nuclear power plant safety and future design issues. The emphasis was on offering exposure to state-of-the-art (single-phase and multiphase) CFD applications reflecting topical issues arising in nuclear power plant design and safety, but in particular to promote the release of high-resolution experimental data to continue the CFD validation process in this application area.

1.1. Scope

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

Nonetheless, emphasis should always be placed on the presentation of new experimental data, especially those relating to two-phase flow, for which sophisticated measurement techniques are required, and for which information is scarce. It is also very important to deepen understanding of the physics before starting a numerical analysis. Experiments providing data suitable for CFD or computational multi-fluid dynamics (CMFD) validation were also welcomed, though these should include local measurements using multi-sensor probes, laser-based techniques (Laser-Doppler velocimetry [LDV], particle image velocimetry [PIV] or laser-induced fluorescence [LIF]), hot-film/wire anemometry, imaging or other advanced measuring techniques for local measurements. It is rapidly becoming an obligation for papers describing experiments to include a discussion on measurement uncertainties.

1.2. Organisational aspects

There were around 120 registered attendees at the CFD4NRS-7 workshop, which was almost the same level of participation as for the previous workshops. The number of Extended Abstracts received for evaluation following the initial announcements was 76. This is similar to previous workshops in the series. All the abstracts were evaluated for suitability by at least two reviewers, and invitations to write a full paper sent out at three hierarchical levels:

- unconditional (favourable reviews had been received from all the reviewers);

- conditional (at least one reviewer was unsure of the final acceptance of the paper);
- guarded (it was anticipated that major revision of the paper would be necessary).

The number of technical papers finally received for evaluation was 56. All technical papers received were evaluated by two to three reviewers, each according to journal standards. Of these, 45 were accepted for oral presentation, and 6 to be presented in poster form. The remaining five papers were withdrawn by their author(s) for various reasons. Five keynote lectures were given, each to introduce the morning/afternoon sessions, as appropriate. In addition, two posters were displayed relating to the OECD-TAMU cold leg mixing benchmark exercise (for which no accompanying paper was requested).

The series of CFD4NRS workshops had rarely been attended by colleagues from China. For this workshop, great efforts were devoted to let domestic colleagues know about the event. Over half of the participants were from domestic universities and institutes. The workshop attracted participants from European countries (15 participants from Germany and 13 from France), which was more than expected. However, the number of participants from Asian countries other than China decreased. The reason for this decrease could have been the conflict with the International Topical Meeting on Nuclear Reactor Thermal Hydraulics, Operation and Safety NUTHOS-12 conference held in October 2018, at Qingdao, China.

A competition had been planned for the poster session to encourage the students to present their work. However, the competition could not be organised due to limited participants in the poster session.

The next workshop in the series, in 2020, will take place at Saclay, France, and Electricité de France has already agreed to organise the event.

In the panel session at the close of CFD4NRS-7, delegates confirmed their interest in attending the follow-up workshop, if possible, and considered the two-year interval between workshops to be appropriate. The two-and-a-half-day duration of formal presentations, with a fieldtrip during the afternoon of the third day, was also accepted as an adequate format.

It was also decided that selected papers from the workshop should constitute a special issue of the Nuclear Engineering and Design (NED) journal, as for previous workshops in this series, and indeed Professor Y. Hassan, General Editor, confirmed this intention. As a consequence, at the conclusion of the workshop, the individual session chairs were approached to declare which of the papers presented in their sessions should go forward for archival recognition in this regard. The evaluation process was subsequently set in motion and the special issue was published[1].

1. Xiong, J., D. Bestion, H. Yassin, N. Sandberg and X. Chai (Eds.) (2020) "Special Section on the '7th Workshop on the CFD for Nuclear Reactor Safety (CFD4NRS-7)'", *Nuclear Engineering and Design*, Volume 358, www.sciencedirect.com/journal/nuclear-engineering-and-design/vol/358/suppl/C#article-15.

2. Technical session summaries from the Co-Chairs

2.1. Session 1: Two-Phase Flow (I)

The session included four papers. The first presented the development and validation methodology of the baseline model for poly-disperse bubbly flow based on multiphase computational fluid dynamics (CFD) in the Euler-Euler framework. The limitations of closure model development based on the integral experimental data was pointed out. Comparison of simulation results with integral experimental data was believed only suited for the validation of the whole model setup in most cases. Two examples were given for development or improvement of closure models based on better understanding and reflection of local phenomena. The first example is updating the bubble-induced turbulence (BIT) model based on the direct numerical simulation (DNS) data. The other was the dedicated experiment to investigate the lateral lift force model. The GENeralized TwO Phase flow (GENTOP) concept was also briefly discussed. The GENTOP concept combines poly-disperse and segregated flows and allows for transitions between the different morphologies. Obtaining CFD-grade data with complete local information and high resolution in time and space demands joint efforts in the community.

The second presentation reported on the development of the wall boiling model for two-phase natural circulation flow in an inclined channel with downward-facing heated wall based on the experiments. The presence of elongated bubbles had been identified and quantified. The long sliding bubbles near the heated wall challenge the classical wall boiling model. Based on the hypothesis that there exists a liquid film between the slug bubble and wall, heat transfer is dominated by liquid film evaporation in the portion occupied by the slug bubbles. A reduction factor is introduced to account for the presence of the slug bubble, while the remaining part is still calculated with the RPI model. The model is implemented into ANSYS CFX and validated with lab- and large-scale data. In the current model the correlation of reduction factor is still very empirical and does not include enough local information. More validation work is required.

The third paper was about CMFD-based departure from nucleate boiling (DNB) prediction in Westinghouse pressurised water reactor (PWR) fuel with four-vane and folded-vane grid spacer designs. A total of eight cases had been analysed using a selected baseline set of STAR-CCM+ boiling/DNB closures. The validation confirms that two-phase CFD can already predict qualitative geometrical effects, but the quantitative accuracy of the predicted DNB heat flux is not sufficient. Selected boiling closures are expected to be most accurate over the range of conditions characterised by bubbly flow. From the current achievement it can be predicted that two-phase CFD can be used to help screen spacer grid design and reduce the need of very costly critical heat flux (CHF) tests.

The fourth paper presented the three-field model to model annular-mist flow in boiling water reactors (BWR), where the liquid film, liquid droplets and the steam are treated as three different fields. The model was validated against experimental data for heated pipes. Even though preliminary good prediction was achieved, more models based on local quantities need to be developed in order to have a predictive tool that can be used for complex geometries. The main obstacle to surpass is the entrainment modelling which is modelled using empirical correlations and large dependency on the droplet diameter. The effects of the droplet diameter on the film mass flow rate is due to the droplet deposition.

2.2. Session 2: Containment (I)

The session included four papers. With the further development of a wall-function approach by the experimental data from the Separate Effect Test for Condensation Modelling (SETCOM) facility, the first paper focused on the turbulent heat and mass transfer in the boundary layer during wall condensation and a detailed validation of a resolved boundary layer (low-Re) CFD approach. Combined with the classical scaling approach based on the friction or shear velocity u_τ (i.e. u+-y+ or T+-y+) and the dimensionless gravity force and the dimensionless heat flux, the wall boundary conditions can be modelled by 3D surfaces instead of a 1D analytical correlation. The 3D models are implemented numerically efficient for u+, T+, Y_S+ and turbulence by means of a non-equilibrium factor and an approximation by radial basis functions into an existing URANS model. The paper was concluded with exemplary discussion of the current model assessment results. It was considered that the models would need to be assessed by further tests.

The second paper presented a three-dimensional numerical simulation of water running down inclined surfaces coupled with a particle wash-off and particle transport model using the open source software OpenFOAM. The water flow on inclined plates is simulated with a two-stage wash-off model for insoluble aerosols based on the Abwaschmodell für unlösliche Aerosole (AULA) wash-off model for aerosols. The simulation results are compared with the experimental data. Resuspension takes place when the wall shear velocity exceeds a critical threshold (Shields criterion). The particle mass that leaves through the outlet is evaluated and compared to the Thermal-hydraulics, Hydrogen, Aerosols and Iodine (THAI) laboratory experiments conducted by Becker Technologies and to calculations done with AULA. Several simulations had been performed with different plate inclinations, initial loading and varying particle properties, such as particle density, particle diameter and wash-off rate. This work will contribute towards the development of a semi-empirical model to predict input parameters for AULA and the creation of new, high-fidelity wash-off models.

In the third paper, a CFD model based on a Eulerian-Lagrangian approach was developed to simulate the nuclear aerosol transport to consider the influence of nuclear aerosol with decay power on the buoyant flows, water-steam balance in the containment and the distribution and flammability of the hydrogen-air-steam mixture. For the Eulerian phase, an Unsteady Reynolds-averaged Navier-Stokes (URANS) approach closed by buoyant k-ω SST model was used. In the Lagrangian phase, drag, gravitational, lift, thermophoresis and turbulent dispersion forces were considered. Additionally, for particles below 1 μm, the effect of Brownian diffusion was also taken into account. The ISP40 STORM experiment SD 11 was used to assess the CFD model. It was found that the default models in OpenFOAM for turbulent dispersion over-predicted the deposition. To allow a first assessment of the effect of decay heat associated with the particles in buoyant flows, a buoyancy driven cavity with Rayleigh number (Ra) of 10^9 was considered. SnO_2 particles with a typical concentration of 1 g/m3 were considered as core melt aerosols. The decay heat was modelled as a volumetric heat source on the particles. It was found that additional heat transfer to the fluid induces local temperature and density changes and this visibly affects also the turbulent transport in stratified flows.

The fourth paper presented considerations on the special case of the Loviisa Nuclear Power Plant which combines the Russian VVER-440 reactor and the western ice condenser containment (ICC). The VICTORIA experimental facility, linearly scaled from Loviisa Nuclear Power Plant containment, was constructed by Fortum to provide the necessary experimental information of the Loviisa ICC thermal hydraulics during small break loss-of-coolant accidents (SBLOCA) and during severe accidents (SA). NRG Petten had previously developed and validated models for containment modelling making use of the ANSYS Fluent CFD code. In a joint research project, Fortum and NRG developed an ice condenser model that was integrated in the CFD containment model of NRG. The model of the VICTORIA facility was constructed for the CFD simulations to validate the complete containment model.

2.3. Session 3: Experiment technique

This session was the only one related to experiments and included three presentations. The first paper dealt with pool scrubbing experiments dedicated to the phenomena in the wet well of a BWR during a severe accident. They were conducted at the THAI facility, which is a versatile medium scale containment test facility. Downcomers – circular, downwards oriented pipes – inject the aerosol laden gas-vapour mixture into the water volume of the pressure suppression chamber. Aerosol scrubbing is relevant in a late phase of the accident when flow rates through these downcomers are low and bubble formation is characterised by little dynamics. The measured decontamination factors for insoluble aerosols were correspondingly low, in the order of magnitude of 10, which is much less than what can be achieved in a Filtered Containment Venting System (FCVS) with optimised injection nozzles. The bubble sizes were measured by high-speed camera observation followed by image processing. The work makes a very important experimental input to severe accident mitigation and modelling applied to BWRs.

The second paper described experiments conducted at an adiabatic 5x5 rod bundle to characterise the flow field downstream of a spacer grid. The paper showed that refractive index matching techniques for the application of optical techniques, like PIV, increasingly become the state of the art in the area of velocity field measurements in bundles. In case of the present paper, PIV was applied to measure a single-phase flow of water at ambient conditions. The paper includes the development of a high-fidelity measurement approach for PIV measurement for cross flow downstream of a spacer grid in a rod bundle. Also, the uncertainty of PIV measurement is quantified, which is very important for CFD validation. In general, the results are very useful for CFD code validation regarding coolant cross flow and cross mixing. Valuable results can be expected from the work at Shanghai Jiao Tong University in this field.

The topic of the third paper was coolant mixing experiments in a vessel with a topological similarity to the pressure vessel of a PWR, although not representing it in a geometrical similarity, using conductance probes and wire-mesh sensors. The authors challenged the scaling rules for mixing experiments regarding deboration and subcooling transients at PWRs. By performing generic experiments in different scales, they planned to obtain CFD-grade measurement data suitable for answering the question of the Reynolds number influence. The first results on time-resolved transport scalar measurements were presented. More was planned to come. The Technical University of Nishny Novgorod (TUNN) has obviously more than one test facility in different scales, which might provide interesting insight into scaling issues in the future. During the discussion there was a debate about the necessity of such a study, since coolant mixing in the RPV was already widely studied in a number of test facilities ranging up to a scale of 1:1 in case of the Upper Plenum Test Facility (UPTF) in Germany. Furthermore, there was experimental evidence from real plant commissioning tests. Still, careful studies are always welcome, and the authors were advised to increase the resolution of their sensors and to better reflect available literature on the issue.

2.4. Session 4: Flow-Induced vibration

There were three excellent presentations about flow-induced vibration (FIV). This phenomenon is analysed by coupling fluid and solid dynamics. It was considered that the usual approach of fluid dynamics is large eddy simulation (LES) because this phenomenon is strongly unsteady. One of the three presentations used LES. However, two of three presentations used Reynolds-averaged Navier-Stokes (RANS) (URANS) and the provided results were very good. If RANS is available, it is more efficient. The general impression was that the vibration frequency was well simulated even though the frequency was shifted from the natural frequency of the solid. However, the damping of vibration was not so well reproduced. This will be studied further.

In the first paper, an advanced numerical framework called NRG-FSIFOAM (finite volume fluid structure interaction solver based on partitioned approach) was presented with the objective to study the dynamics of Turbulent Induced Vibration (TIV) and to use if for the application of nuclear reactor safety (NRS). The NRG-FSIFOAM is based on the use of partitioned algorithms for the fluid and solid problems coupled by means of the state-of-the-art approach for the simulation of strongly coupled FSI problems. To overcome the default defect of URANS models which are not capable of modeling the fluctuations at the smaller scales, an innovative Pressure Fluctuation Model (PFM) was implemented in NRG-FSIFOAM with the objective of complementing the average field computed with the URANS approach with a stochastic model that is able to reproduce the chaotic fluctuations of the turbulent field. This numerical framework was first validated in good agreement with strongly coupled FSI benchmark cases. The capabilities of the PFM were then assessed for a nuclear application case for which experimental measures are available. It was found that the PFM is able to reproduce the dynamics of the vibrations observed in the experiments while the classical URANS model fails to predict physical oscillations.

The second paper described how the codes ANSYS CFX-MOR and ANSYS CFX–ANSYS Mechanical are validated against the Vattenfall Rod Vibration Experiment data. The experimental setup consisted of a Plexiglas test section with a slender stainless steel rod in the middle, which was pulled and then released. The calculated time dependent vibration amplitude was compared with measured data for the water and air test cases. The analyses showed that the nature of the vibrations for the cases with flowing fluid was well predicted, while underestimations of the vibration amplitude and phase shift were observed in the cases with stagnant flow.

The third paper presented results of numerical and experimental activities to validate the calculations of flow-induced vibrations of equipment flexible structures. A geometrically simple model of a bluff body which represents two flexible cylinders arranged in-line in the working medium cross flow was taken as a subject of research. Modes with the Reynolds numbers Re of 10^4 were studied. Hydrodynamics was modelled using an LES model for the turbulent flow. The validation showed that the hydrodynamic calculation and calculation of the cylinder vibration levels at the vortex shedding frequency are in satisfactory agreement with the available experimental data. However, the coupled calculation of vibration levels provides overassessment for the natural frequency. The obtained results testify to the conservative nature of the applied approach to calculate vibration amplitudes of flexible bodies.

2.5. Session 5: Bundle flow

Three papers were presented in session five for fluid flow and heat transfer of single phase in rod bundle. Two papers used the commercial CFD code, STAR-CCM+ and the third paper developed in-house code using the OpenFOAM library. The turbulence models used in these papers are the realisable k-ε model, PANS (Partially Averaged Navier-Stokes) k-w model, k-w SST model. The CFD simulations are verified in all three papers but the validation process was performed only in one paper. It is impressive to make an effort for multiphysics coupling and application of high-end turbulence model.

The first paper described research on the particle behaviour and cladding oxidation in a PWR fuel bundle by CFD methodology to improve conclusions for design optimisation. The movement of discrete particles in the moderator can already be simulated by a Lagrangian model with the particle tracking approach. A reference 3D PWR corrosion model was started to be developed, which can calculate local parameters on a fuel rod surface 2D model to predict oxide thickness and component deposition taking into account the mixing effects during the moderator transport. The calculated 2D surface coolant temperatures were coupled to a dedicated fuel rod corrosion model, covering the full fuel rod in azimuthal and axial direction.

In the second paper, a PANS model was used. This is an energy based, filtered representation of the Navier-Stokes equations. As part of the bridging models paradigm family, it is possible with such a model to resolve the turbulent space and time scales of interest. The most important large-scale unsteadiness can be resolved at a minimal computational cost. The model and its solution verification had been tested for a turbulent flow inside a 5x5 fuel bundle, with a single spacer grid and split-type mixing vanes. The results will be compared with particle image velocimetry data available in literature.

The third paper described how the local heat transfer behaviour of wire-wrapped rod bundle in liquid metal cooled reactors, can be studied with OpenFOAM to calculate the detailed temperature and heat flux distribution on the surface of rod and wire. In the study, two CFD models were created for a bare rod bundle and a wire-wrapped 19-rod bundle. For the bare rod model, a periodical boundary condition was used to simulate the heat transfer of developed flow. For the wire-wrapped 19-rod bundle case, the normal inlet outlet boundary was imposed to the fluid region with a length of two wire pitches. In the CFD studies, the models were divided into solid and fluid regions. The k-omega SST model was used for the turbulent simulation. The simulation result showed a cosine like local heat transfer distribution in the circumferential direction of the rod.

2.6. Session 6: Mixing (I)

There were three presentations in the session. The first addressed the verification and validation in LES of a triple parallel jet flow. In the verification part, the numerical error was first estimated with two LES on different grids where no SGS model was set. It was shown that the kinetic energy was mostly overestimated in the mixing region of the flow, leading to a negative effective SGS kinetic energy. Then, the contribution of the SGS model was investigated. The order of convergence of the SGS kinetic energy with the Smagorinsky model was found to be in good agreement with other results. Moreover, the WALE model was found to converge more rapidly towards the total kinetic energy than the Smagorinsky model. The LES on the coarsest mesh had a lower discrepancy with experiment than those on the finest mesh; the origin of this unexpected results needs to be investigated.

The second presentation was about water mixing during In-Containment Refueling Water Storage Tank (IRWST) heatup for the AP1000® Plant. The CFD results show that ANSYS CFX can simulate the flow and heat-transfer characteristics of natural convection in the IRWST, and the error of the simulation was acceptable. With thermal phase change included in the model, the results matched the test data better than that without thermal phase change, especially later in the test when the IRWST was at elevated temperatures, when subcooled boiling is more likely to occur. Finally, both the CFD results and hot functional testing (HFT) data show that the temperature distributions in the IRWST are horizontally stratified.

In the last presentation, the turbulent natural convection formed in a rectangular enclosure with a curved surface heated from below was shown with experimental and numerical methods. The dependency of the Nusselt number on the Rayleigh number is quantitatively investigated. It was found that the Nusselt number is proportional to the Rayleigh number and the exponent is larger than the values found in the literature. The distribution of fluid velocity and turbulent kinetic energy in the region near the curved surface was also obtained with the PIV method. Comparing against the experimental data, it was found that the presented numerical data underestimate the wall temperature profile over the heating section and the v2-f turbulence model with three different turbulent flux models fails to reproduce the wall temperature. It was also found that the predicted velocity distribution near the heating section is much larger than the experimental data. The discrepancy of the wall temperature between the numerical and experimental results can be attributed to the overestimation of fluid velocity and the components of turbulent heat fluxes.

The session was well attended and showed that turbulent mixing processes are still valuable research-objectives in the nuclear community.

2.7. Session 7: Liquid metal

There were four presentations in the session. The first paper was about CFD analysis of the flow field in the MICAS experiments (mock-up of the hot plenum ASTRID reactor project) at CEA which represents a water model of the hot pool of the Astrid reactor. Trio_CFD was used to perform the RANS computation and the obtained results, even with the use of a linear k-epsilon model, were found to be qualitatively in good agreement with the experiments.

The second paper evaluated the pressure drop in a fuel bundle and reflector of the fuel assembly for a prototype Gen-IV sodium-cooled fast reactor (PGSFR) using computational fluid dynamics (CFD) and analytical methods. The commercial CFD codes, ANSYS CFX and STAR-CCM+, were used to predict the pressure drop in a fuel bundle and reflector. The RANS turbulence models used in this CFD analysis are the standard k-e model, Realizable k-epsilon model and SST k-omega model. The CFD calculation and correlations predicted the bundle friction factor, which agrees well with the experimental data.

The third paper investigated the influence of turbulence modelling on the heat transfer, especially for fluids with low Prandtl numbers like sodium and lead-bismuth eutectic (LBE). A detailed CFD model was prepared and presented. Special attention was given to optimise the mesh for accurate geometry representation and fast convergence. Simulations were run with three different turbulent Prandtl numbers (0.9, 1.5 and variable values generated by a look-up table method). It was found that the influence of the choice of Pr_t is low for fluids with very small Pr numbers like sodium, while it is of much higher importance for fluids with a lower thermal conductivity. The conjugate heat transfer has a small impact at the touching zone between rod and wire.

In the fourth presentation, a CFD calculation was performed to validate the design of a reactor flow distribution test facility for the PGSFR. The porous media approach was adopted to simulate the core region, which has complex geometries of 313 fuel assemblies for computational efficiency. The full velocity and pressure distribution in the test vessel were acquired from a CFD analysis. The flow rate and pressure drop of the fuel assemblies and the IHX were compared with the target values suggested as the reference parameters, and the vessel-wise velocity and pressure distribution were observed for surveying any defective flow in a vessel. From the results of the CFD analysis, it was confirmed that the global flow behaviour in the test vessel was hydrodynamically acceptable.

2.8. Session 8: Two-Phase flows II

This session consisted of four papers. The first three were loosely related, dealing with different aspects of gas-liquid flows, while the fourth paper dealt with solid-liquid phase change.

The first paper addressed improved turbulence modelling for a rising Taylor bubble. Specifically, a damping source term was added to RANS k-Epsilon and k-Omega models, which resulted in significantly improved agreement with experimental data and high-resolution LES simulations.

The second paper discussed improved interphase heat transfer coefficient (HTC) correlations for Euler-Euler closures. A review of existing interphase HTC correlations was presented, an improved HTC correlation was proposed, and results using this new correlation were shown for steam condensation in subcooled water, subcooled wall boiling and flashing nozzle flow. The results were very encouraging and some suggestions for further development were discussed.

The third paper described sensitivity studies of different boiling interfacial momentum closures, including drag, lift and wall lubrication force models. There was a good discussion with the audience related to the experimental data used for comparison and assumptions made about the initial bubble size distribution used in the simulations.

Finally, the fourth paper was on numerical modelling of frozen walls in a molten salt reactor. The CFD simulations were coupled to neutronics predictions for the heat sources, and the frozen salt in the CFD simulations was captured by a temperature-dependent porosity. Preliminary results show that maintaining a frozen salt wall layer may be a challenging design problem.

The papers in this session demonstrated the range of models and applications of two-phase CFD simulations, as well as their associated challenges.

2.9. Session 9: Containment (II)

The Containment II session contained three presentations related to analysis and validation of different modelling challenges related to three aspects of the hydrogen issue: mixing, mitigation and combustion.

The first paper addressed the assessment of a Passive Autocatalytic Recombiner (PAR) model based on the manufacturers' correlation and implemented in Fluent The main challenges were represented by the flow conditions, i.e. the flow field around the PAR was opposing the buoyant chimney flow cheated by the PAR. Experimental results obtained within the multi-compartment THAI+ facility during the NEA THAI-III project were used as experimental reference. Both simulation and experiments revealed that the imposed counter-current flow did not affect the PAR performance once the PAR was operating. The level of agreement between measurement and simulation indicates that the relatively simple correlation based model for the recombination rate can be applied without introducing significant modelling errors, even under the given challenging boundary conditions.

The second paper addressed the systematic assessment of an OpenFOAM-based turbulent mixing model against a complete experimental test series on the erosion of a helium stratification by means of a vertical jet, conducted in the PANDA facility within the NEA SETH-2 project. The nine different cases consider variation of the density gradients, gas mixture and injection locations. Even though the results did not reveal a fundamental disagreement with the experiments, several inconsistencies related to the mixing rate or heat transfer (gas temperature level) were identified in the thorough assessment of all different experimental tests. Different variations, e.g. of turbulence model constants or inclusion of thermal radiation modelling, suggest it would be good to take a closer look at these issues and create specific guidelines or modelling standards for this type of problem.

The last paper investigated the effect of the initial turbulence level on the downward propagation of a slow flame on basis of a deflagration experiment conducted within the NEA THAI project. The GASFLOW-MPI code was used in the systematic parametric analysis to assess the counteracting effects of thermo-diffusivity instability and higher initial turbulence level which promote flame propagation and the Rayleigh-Taylor instability and buoyancy, which tend to suppress the downward flame propagation. The work encourages the inclusion of the instabilities and turbulence intensity into the formulation of the flame velocity. On the experimental side, the analysis encourages the measurement of the initial turbulent condition in combustion experiments.

Containment type simulations can be characterised by large computational domains and long transients, which usually prohibited repetitive analysis runs to address sensitivities. It can be noted that all presented work was based on multiple simulations, conducted either to quantify mesh sensitivity to different initial and boundary conditions (scenarios / test cases) to base model assessment on a broader basis. Looking at the modelling approaches, there was a harmonisation but still no complete consensus about the physics to be included for containment typical analysis (e.g. gas radiation) about model coefficients (e.g. turbulence transport). This should be addressed in more specific guidelines on containment flows to generalise the experience obtained for individual model assessments.

2.10. Session 10: Mixing (II)

There were four presentations in this session. The first presented the analysis of the thermohydraulic behaviour in the cold legs and downcomer of a pressurised water reactor primary circuit during an intermediate-break loss-of-coolant accident scenario located on one of the cold legs. First, a physical analysis of the accidental scenario was proposed, including a Phenomena Identification and Ranking Table analysis to comprehend the modelling complexity on an industrial geometry. Second, a presentation of NEPTUNE-CFD was given, including a discussion on the physical models that will be used to deal with this accidental scenario.

In the second presentation, large eddy simulation (LES), using code NEK5000, of an experimental cold leg benchmark investigating buoyant mixing among two fluids was developed for verification and validation. It was concluded that instabilities due to resolving more of the interface using the molecular Schmidt number were essential for accurately predicting this type of flow.

In the third presentation, a recently published pressure-based compressible multiphase flow model was validated for applications relevant to nuclear safety, such as flashing of high-pressure water through valves and nozzles and steam jets into suppression pools. Good matches with the experimental results were observed, thus validating the compressible multiphase flow model for critical scenarios in nuclear safety.

In the last presentation, through the comparison of the simulated results with the results performed in the THAI facility, it was found that STAR-CCM+ 11.02 with the fluid film model simulating the steam condensation predicted the steam concentration, gas temperature, and vessel wall temperature with an uncertainty range of approximately 20%. This difference between test and simulation might be caused by radiative heat transfer between the steam-air gas mixture and the inner wall, which is not considered in the CFD simulation. Therefore, it was proposed that the radiative heat transfer model be included in the best practice guidelines (BPGs) to accurately simulate the natural convection flow induced by the steam condensation.

2.11. Session 11: Pressurised thermal shock

In this session, three papers were presented. The composition of this session was optimal to demonstrate a full line of efforts to solve a safety-relevant problem by means of CFD simulations. The first paper gave an overview of calculations performed in the frame of the PTS benchmark based on ROCOM data. The injection of a deborated plug with and without density difference was discussed, both by showing the experimental results and by comparing them to CFX calculations. The achieved good performance of the code demonstrates in an impressive way that CFD is mature for this kind of single phase mixing applications, of course under the conditions outlined by the Best Practice Guidelines.

The application of this achievement for the simulation of the bounding subcooling transient in a German type PWR was presented in the third paper. This paper showed it was possible to demonstrate that simplified models based on experimental evidence obtained at the full-scale test facility UPTF provide conservatisms and the CFD simulations were able to uncover additional safety margins regarding the subcooling of the RPV wall during the bounding loss-of-coolant-accident (LOCA) scenario.

The second paper dealt with DNS related to the PTS issue. The NRG in Petten, in the Netherlands, conduct DNS simulations in a simplified downcomer geometry with the aim of generating high-quality data for the validation of turbulence models. The idea behind this is the indirect transfer of knowledge from DNS calculations to the real reactor scale by validated RANS or LES models. DNS was successfully applied to a case without buoyancy and the result of this numerical experiment is available for code validation. The next step concerns a test case with density difference, which causes a considerable increase in complexity. Some more simplifications of the geometry will be necessary. Still, an increase of the size of the needed mesh – up to 1.5 billion cells – seems to be necessary and a sufficiently powerful computer hardware has to be found.

With only three presentations, the session was able to give a comprehensive picture of the efforts needed to solve a safety relevant problem using CFD. From fundamental DNS analyses allocated at the front end of the CFD validation chain (paper #2) via the already quite applied research at ROCOM and the related code validation exercises (paper #1) up to the full-scale experiment (UPTF) final application to real plant conditions (paper #3), a practically complete picture was given. This demonstrates the success of the CFD4NRS movement.

2.12. Session 12: Plant application (I)

This session had three papers for the CFD application to hot leg with Automatic Depressurization System (ADS) in AP1000, AES-2006 reactor pressure vessel (RPV) simulating VVER1200, and upper plenum/hot leg for AP1000. The three papers considered single-phase fluid and two-equation models for turbulence. The STAR-CCM+ code was used to simulate the fluid flow and heat transfer in hot leg with ADS and AES-2006 RPV. The in-house code Code-Saturne was applied to predict flow and temperature distribution of coolant in upper plenum and hot leg of AP1000. The three papers performed the verification analysis but only one paper provided the validation result. The CFD predictions are quite reasonable and showed a positive outlook for the application of CFD methods in the optimal design of nuclear power plants.

The first paper used CFD to assess the susceptibility of vibration of an AP1000® reactor ADS pipe where it intersects with the hot leg nozzle piping. These CFD results were used in a qualitative manner to guide proposed design modifications and subsequent scale model testing. Two types of CFD analyses were performed to fulfil different purposes: (i) steady analyses were performed to provide an indication of flow features such as separation points and shear layer structure; (ii) unsteady simulations were then performed on the baseline and final designs to further validate the final design selection.

In the second paper, a virtual model of the primary circuit at nuclear power plants with VVER reactors was developed by CFD methodology to prove basic technical solutions of AES-2006 RPV design concerning questions of hydrodynamics and heat exchange. The results of calculations using elaborated models are three-dimensional distribution of temperature, pressure, velocity vector of the coolant and other values, characterising heat and mass transfer in the flow path of RPV primary circuit. The correspondence of the calculation data to design data is shown by the results of the comparison within the range of technical tolerances and errors and that proves the correctness of the simulation.

The third paper conducted CFD analysis of PWR upper plenum thermal-hydraulic phenomena with CODE_SATURNE. To make a comparison between different geometries for future research, a simplified AP1000 upper plenum geometry was adapted to a 1/5 scaled four-loop BORA (RPV mock-up at 1/5 scale) mock-up facility built by EDF R&D to carry out a CFD simulation. The calculation domain includes the upper half core, upper plenum and four hot legs. The geometry and mesh were generated by Salome and the calculation was conducted using the Code_Saturne developed by EDF. A uniform flow rate and a given temperature field distribution are imposed to the upper half core inlet and outlet, respectively. Three-dimensional coolant temperature and flow fields at the core outlet level and in the upper plenum and hot legs were obtained and discussed. The suction effects of hot legs on the flow fields in the upper plenum were achieved. Also, the temperature field evolution in hot lets was analysed. This work is meaningful for the deep understanding of the thermal-hydraulic phenomena in the PWR upper plenum.

2.13. Session 13: UQ and coupling

This session included two papers. The first provided the results of an Uncertainty Quantification (UQ) method for CFD validation for turbulent mixing experiments from

GEMIX (mixing experimental facility [PSI]). NRG participated in this benchmark with a UQ method based on the ASME Verification and Validation (V&V) standard for UQ in CFD. The propagation of the uncertainty in the input parameters was sampled with the Latin Hypercube Sampling. The numerical uncertainty was determined by means of Richardson extrapolation. A source of uncertainty which was not evaluated explicitly was that of the turbulence model input parameters, which depend heavily on the flow conditions. In order to investigate this contribution, the turbulence model parameters have been considered as uncertain input variables. The results of this so called extended ASME methodology were presented and discussed. It was found that the uncertainty in the turbulence model input parameters has a significant contribution to the overall uncertainty in the simulation results for the considered GEMIX mixing experiment.

The second paper described the development and validation of the multiscale multiphase, numerical platform NEPTUNE CFD/CATHARE. The coupling between the codes CATHARE and NEPTUNE CFD was presented in the context of single-phase flows. A verification test case was first provided to showcase the correct implementation of the coupling. Finally, a validation test case with strong mixing effects based on a double T-junction experimental facility was studied.

2.14. Session 14: Plant applications (II)

This session included two presentations. The first concerned thermal-hydraulic analysis of a reactor plant for an icebreaker ship. Coupled 1-D/3-D simulation was performed to predict temperature at the core inlet, due to the concern that incomplete mixing could affect thermocouple readings used to control the plant. To accurately capture the mixing effects, LES was used for the CFD part of the simulations. Verification results for the code coupling technology were also presented.

The second paper addressed the thermal control temperature predictions at the fuel assembly outlets for a VVER-1000 reactor. Inlet boundary conditions for the simulations were provided by a sub-channel code. Results were compared to measurements, showing good agreement. After further validation, it will be possible to predict thermocouple readings in the hot legs as well.

These papers show the challenges of real-world plant applications of CFD, which include complex geometries, very large models and complex physics, as well as some of the solutions such as coupling to system and sub-channel codes.

Annex A. Keynote lectures

1. Guo-Han Chai; Nuclear and Radiation Safety Center (NSC), China
   Application of CFD in Nuclear Safety License in China
2. Horst-Michael Prasser; ETH Zürich, Switzerland
   High-Resolution Measurements in Two-Phase Flow Experiments in Adiabatic and Heated Rod Bundle and Sub-channel Models
3. Dominique Bestion; CEA, France
   Requirements for CFD-grade Experiments for Nuclear Reactor Thermal-Hydraulics
4. Yassin Hassan; TAMU, United States
   OECD-TAMU Cold Leg Mixing Benchmark Exercise
5. Seiichi Koshizuka, University of Tokyo, Japan
   Guidelines for Verification and Validation for Engineering Simulation

Annex B. Poster papers

1. S. K. Bikezin, D. A. Oleksyuk

   CFD Modelling of the 37 Rod Bundle Experiment with Different Spacing Grids

2. Chen Chong, Wang Mingjun, Tian Wenxi, Qiu Suizheng, Su Guanghui, Qi Yubo

   Simulation of "Dead Legs" Phenomena for PWR Nuclear Power Plant

3. A. Graževičius, A. Kaliatka, E. Ušpuras

   Numerical Study of the Natural Convection and Thermal Stratification Phenomena in a Rectangular Enclosure with a Horizontal Heat Source

4. Y. Bouaichaoui, T. Höhne

   CFD Benchmark Study of Pressurised Thermal Shock Experiment of the ROCOM Test Facility

5. A. S. Noskov, A. A. Falkov, D. L. Shipov

   STAR-CCM+ Verification and Application for a Reactor Core Fluid Dynamics Analysis

6. Ji Wen-ying, Zong Wei-xin, Lv Yi-jun, Luo Han yan, Lu chang dong, Yang Jiang

   CFD Analysis of the Inlet Blockage Accidents for Single Fuel Assembly of the LBE-Cooled Fast Reactor

7. Yazhe Lu, Fan Yang, Zhiqiang Zou, Huanhuan Peng, Jian Deng

   CFD Related Studies on Hydrogen Issue in Nuclear Power Plant

8. Luguo Liu, Zhongchuan Li, Sijia Du, Songwei Li, Xi Chen, Peiying Li, Yu Liu

   CFD Application on Fuel Assembly Design in NPIC

9. Yingzi, Zhu, Jinbiao Xiong

   Development of MPS-Based Eutectic Reaction Model for Severe Accident Simulation

10. Mubashir Hassan, Jinbiao Xiong, Xu Cheng

    CFD validation and sensitivity analysis of OECD-TAMU cold leg mixing benchmark

Annex C. Technical programme (presented orally)

Session 1: Two-Phase flow (I)

Co-Chairs: D. Bestion (CEA, France), J.-P. Simoneau (EDF, Germany)

1. D. Lucas, E. Krepper, Y. Liao, T. Höhne, R. Rzehak, F. Schlegel, T. Ziegenhein

   Multi-Fluid Models for Gas-Liquid Flows

2. Hyoung-Tak Kim, Dae-Kyum Lee, Kwang-Hyun Bang

   Wall Boiling Model for Flow Boiling in an Inclined Channel with Downward-Facing Heated Wall

3. Robert A. Brewster, Emilio Baglietto, W. David Pointer

   Assessment of Boiling Models for CFD-Based Prediction of DNB on Westinghouse PWR Fuel Grids

4. Salvatore Raddino, Yann Le Moigne, Tobias Strömgren, Jean-Marie Le Corre

   Modelling of Annular Flow with a Three-Field Approach

Session 2: Containment (I)

Co-Chairs: Y. Hassan (TAMU, United States), J. Xiao (KIT, Germany)

1. S. Kelm, H. Müller, A. Hundhausen, C. Druska, A. Kuhr, H.-J. Allelein

   Development and Validation of a Multi-Dimensional Wall-Function Approach for Wall Condensation

2. K. Amend, M. Klein

   Modelling and Simulation of Wash-Down of Fission Products by Water on Containment Walls

3. M. Kampili, S. Kelm, H.-J. Allelein

   Modelling of Aerosol Transport and Decay Heat Distribution in Containment Flows

4. T. Rämä, T. Toppila, D. Visser, A. Siccama

   Validation of the Containment Analysis CFD Model for VVER-440 Type Containment with Ice Condenser

Session 3: Experiment technique

Co-Chairs: H.-M. Prasser (ETH Zürich , Switzerland), J. B. Xiong (SJTU, China)

1. M. Freitag, B. von Laufenberg

   Measurement and Evaluation of the Bubble Dynamics in the Vicinity of a Downcomer under Conditions Typical for Pool Scrubbing in the Wet-Well of BWR

2. W. Qu, S. Chen, J. Xiong, X. Cheng

   Experimental Measurement of Turbulent Flow in a 5×5 Rod Bundle with Mixing Vane Spacer Grids for CFD Validation

3. M. A. Bolshukhin, A. V. Budnikov, A. A. Barinov, D. N. Patrushev, S. M. Dmitriev

   Experimental Studies on the Turbulent Flow Steady-State Mixing Processes in the Large-Scale Test Facility

Session 4: Flow-Induced vibration

Co-Chairs: Y. H . Yang (SJTU, China), S. Koshizuka (University of Tokyo, Japan)

1. S. Sharma, D. De Santis, A. Shams

   An Advanced URANS Solver to Predict Turbulence Induced Vibrations in Nuclear Reactor Applications

2. A. Papukchiev

   Numerical Analysis of Reactor Relevant Vibrations Using Advanced Multiphysics CFD-CSM Methods

3. M. A. Bolshukhin, AV. Budnikov, E. I. Shmelev, D. A. Kulikov, A. I. Patrusheva

   Approaches to Modelling of Hydrodynamic Forces to Calculate Flow-Induced Vibrations of Equipment Flexible Structures

Session 5: Bundle flow

Co-Chairs: W. Tian (XJTU, China), W. K. In (KAERI, Korea)

1. D.-Y. Sheng, M. Seidl

   Numerical Investigation of Particle Behavior and Cladding Oxidation in a PWR Fuel Bundle by CFD Methodology

2. G. Busco, Y. A. Hassan

   Solution Verification of PANS Modelling for a 5x5 PWR Fuel Assembly

3. X. Wang, X. Cheng

   A CFD Study of Local Heat Transfer in Triangular Arrayed Rod Bundle

Session 6: Mixing (I)

Co-Chairs: G.-H. Chai (NSC, China), T. Höhne (HZDR, Germany)

1. P.-E. Angeli

   Verification and Validation in LES of Triple Parallel Jet Flow for a Thermal Striping Investigation

2. H. Xu, R.F. Wright

   Water Mixing During IRWST Heatup under Natural Circulation for the AP1000® Plant

3. X. Chai, B. Chen, J. Yao, X. Liu, J. Xiong, X. Cheng

   Numerical Simulation of Heat Transfer Properties over a Curved Heated Surface in an Enclosure

Session 7: Liquid metal

Co-Chairs: D. Bestion (CEA, France), A. Shams (NRG, Netherland)

1. U. Bieder, J. Maillard, Y. Gorsse, D. Guenadou

   CFD Analysis of the Flow in the MICAS Experimental Facility, a Water Model of the Hot Pool of a Sodium Cooled Fast Reactor

2. W. K. In, K.-G. Lee, S.-R. Choi, J.-S. Cheon

   Evaluation of Pressure Drop in Fuel Bundle and Reflector for Sodium-Cooled Fast Reactor

3. M. Böttcher, R. Krüssmann

   CFD Simulation of Liquid Metal Flow in a 19 Rod Wrapped Wire Assembly with Focus on Turbulent and Conjugate Heat Transfer

4. W. S. Kim, S.-K. Chang, D.-J. Euh

   CFD Calculation for Preliminary Analysis of SFR Reactor Flow Distribution Test

Session 8: Two-Phase flow (II)

Co-Chairs: D. Lucas (HZDR, Germany), R. Brewster (Westinghouse, United States)

1. E. M. A. Frederix, A. S. Hamraz, J. G. M. Kuerten, E. M. J. Komen

   Simulation of a Rising Taylor Bubble Using Improved Reynolds-Averaged Models

2. Y. Liao, E. Krepper, D. Lucas

   A Baseline Closure Concept for Simulating Bubbly Flow with Phase Change: Interfacial Heat Transfer Coefficient

3. D. Jin, J. Xiong, X. Cheng

   Effects of Interphase Force Models and Turbulence Model on Two-Phase Flow in Vertical and Inclined Upward Pipe

4. G. Cartland-Glover, A. Skillen, D. Litskevich, S. Rolfo, D. R. Emerson, B. Merk, C. Moulinec

   On the Numerical Modelling of Frozen Walls in a Molten Salt Fast Reactor

Session 9: Containment (II)

Co-Chairs: S. Kelm (FZJ, Germany), X. Chai (SJTU, China)

1. Y. Halouane, A. Dehbi

   CFD Simulations of Passive Autocatalytic Recombiner Operation inside the THAI+ Two-Compartment Facility

2. R. Krpan, I. Kljenak

   Simulation of PANDA SETH2 Experiments on Containment Atmosphere Mixing Caused by Vertical Injection

3. J. Xiao, M. Kuznetsov

   Effects of Initial Turbulence Intensity on Downward Hydrogen Flame Propagation in THAI HD-23 Test

Session 10: Mixing (II)

Co-Chairs: T. Höhne (HZDR, Germany), A. Shams (NRG, Netherland)

1. N. Mérigoux, J. Laviéville

   Application of NEPTUNE_CFD to Intermediate-Break Loss of Coolant Accident in the Downcomer Side of a Pressurised Water Reactor

2. J. K. Lai, E. Merzari, Y. A. Hassan, P. Fischer, O. Marin

   Verification and Validation of Large Eddy Simulation with Nek5000 for Cold Leg Mixing Benchmark

3. C. Narayanan, D. Lakehal, A. D. Carlson, L. Zhang, K. Fu

   Application of Pressure-Based Compressible Multiphase Flow Approach to Flashing in High-Pressure Valves and Under Expanded Gas Jets into Suppression Pools

4. H. S. Kang, R.-J. Park, D. H. Kimand M. Freitag

   CFD Analysis for Dissolution of a Steam-Air Stratification by Natural Convection in the THAI Facility

Session 11: Pressurised thermal shock

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

1. T. Höhne, S. Kliem

   Numerical Simulation of the IAEA Benchmark Regarding ROCOM PTS Test Cases

2. S. Aggarwal, A. Shams, D. De Santis, E. M. J. Komen

   Design of a Single-Phase PTS Numerical Experiment with Buoyancy Effects for a Reference DNS

3. I. Cremer, R. Trewin, S. Grams, A. Mutz

   Two-Phase Pressurised Thermal Shock Analysis with CFD Including the Effects of Free-Surface Condensation

Session 12: Plant application (I)

Co-Chairs: W. K. In (KAERI, Korea), W.X. Tian (XJTU, China)

1. R. A. Brewster, T. A. Bissett, G. A. Meyer, T. K. Meneely

   CFD Analyses of theAP1000® ReactorHot Leg/Automatic Depressurization System (ADS) Nozzle

2. A. P. Skibin, V. Y. Volkov, L. A. Golibrodo, A. A. Krutikov, O. V. Kudryavtsev, Y. N. Nadinskiy

   Development of the Virtual CFD model of VVER NPP Primary Circuit

3. L. Wang, M. Wang, T. XU, J. MIN, W. Du, W. Tian, S. Qiu, G. Su

   CFD Analysis of PWR Upper Plenum Thermal Hydraulic Phenomena with CODE_SATURNE

Session 13: UQ and coupling

Co-Chairs: S. Koshizuka (University of Tokyo, Japan), D. Bestion (CEA, France)

1. A. Cutrono Rakhimov, D. C. Visser, E. M. J. Komen

   Uncertainty Quantification Method for CFD Validated for Turbulent Mixing Experiments from GEMIX

2. C. Koren, C. Geffray

   Progress in the Development and Validation of the Multiscale, Multiphase, Numerical Platform NEPTUNE_CFD/CATHARE

Session 14: Plant applications (II)

Co-Chairs: T. Höhne (HZDR, Germany), R. Brewster (Westinghouse, United States)

1. M. Bolshukhin, A. Budnikov, D. Sveshnikov, R. Romanov

   Experience with the Practical Application of the 1D-3D Coupled Thermal-Hydraulic Analysis to Validate Future Operating Modes of the RITM-200 Reactor Plant for the New-Generation Nuclear-Powered Icebreaker

2. D. A. Oleksyuk, V. A. Bugayeva, D. R. Kireeva

   Simulation of Standard Temperature Control Indications at the Outlet of a Fuel Assembly of the VVER 1000 Reactor of Rostov NPP Unit No. 2