Nuclear Energy Agency

NEA/CSNI/R(2022)2

Unclassified **English text only**

28 July 2023

NUCLEAR ENERGY AGENCY
COMMITTEE ON THE SAFETY OF NUCLEAR INSTALLATIONS

Computational Fluid Dynamics for Nuclear Reactor Safety (CFD4NRS-8) Workshop, 25-27 November 2020

This document is available in PDF format only.

JT03523756

Committee on the Safety of Nuclear Installations (CSNI)

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

The Committee constitutes 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 serve 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 considers the safety implications of scientific and technical developments of future reactor technologies and designs as well as human and organisational research activities and technical developments that affect nuclear safety.

Foreword and acknowledgements

The Nuclear Energy Agency (NEA), the Committee on the Safety of Nuclear Installations (CSNI) and the Working Group on Analysis and Management of Accidents (WGAMA), acknowledge the significant contributions of the individuals who played a key role in the preparation of this summary report, and those who had a leadership role in the conduct and success of the virtual workshop on computational fluid dynamics for nuclear reactor safety (CFD4NRS-8), including the chair, Didier Banner (EDF), and co-chair, François Barré (IRSN), the organising committee, which is composed of William Benguigui (EDF, IMSIA), André Bergeron (CEA), Guillaume Bois (CEA), Anthony Dyan (EDF), Mathieu Guingo (EDF), Nicolas Mérigoux (EDF), Pierre Moussou (EDF, IMSIA) and Maria-Giovanna Rodio (CEA). Additional thanks are extended to all the chairs and the computational fluid dynamics (CFD) task group leaders for preparing this summary and to C. Richardson, C. Tephani-M'Pania, I. Ryan and Y. Rivera for their editorial support.

Leading author

Mathieu Guingo	Électricité de France, France

Contributors

Guillaume Bois	Commissariat à l'énergie atomique et aux énergies alternatives, France
William Benguigui	Électricité de France, France
Romain Denefle	Électricité de France, France
Anthony Dyan	Électricité de France, France
Philippe Freydier	Électricité de France, France
Chai Koren	Électricité de France, France
Nicolas Mérigoux	Électricité de France, France
Stéphane Mimouni	Électricité de France, France
Franchine Ni	Électricité de France, France
Jan-Patrice Simoneau	Électricité de France, France
Jean-François Wald	Électricité de France, France
Thomas Höhne	Forschungszentrum Dresden-Rossendorf, Germany
Stephan Kelm	Forschungszentrum Juelich, Germany
Dirk Lucas	Forschungszentrum Dresden-Rossendorf, Germany
Pierre Ruyer	Institut de Radioprotection et de Sûreté Nucléaire, France
Evgeny Shmelev	JSC Afrikantov OKBM, Russia
Jinbiao Xiong	Shanghai Jiao Tong University, China
Martina Adorni	Nuclear Energy Agency

This report was approved by the CSNI in December 2021 and prepared for publication by the NEA Secretariat.

Table of contents

Executive summary **6**

List of abbreviations and acronyms **8**

1. Workshop objectives and organisation **10**

1.1. Scope and history of the CFD4NRS workshop series 10
1.2. Main figures of the 8th edition 11

2. Committees and acknowledgements **12**

3. Technical contents of the workshop **14**

3.1. Keynote sessions 14
3.2. Technical sessions 15
3.3. Conclusion 18
3.4. Technical sessions continued 18

Executive summary

The eighth instalment of the workshop on computational fluid dynamics for nuclear reactor safety (CFD4NRS-8) was organised by the Nuclear Energy Agency (NEA) Committee on the Safety of Nuclear Installations (CSNI) and held virtually on 25-27 November 2020. It aimed to foster exchanges between specialists on the application to nuclear safety problems of the latest advances in 3D numerical tools, as well as on new experimental data for validation purposes. It was initially scheduled to take place at the EDF Lab Paris-Saclay, France, but took place as an online event due to the worldwide COVID-19 outbreak. The salient points of the workshop are as follows:

- **New interesting experimental data** with innovative techniques were presented in two dedicated sessions showing, for instance, new insights into the physics of the departure from nucleate boiling (DNB) by the Massachusetts Institute of Technology (MIT) team; or the application of the magnetic resonance velocimetry (MRV) method to obtain local information on turbulent single-phase flows. This "CFD-grade" data will certainly drive future improvements in the CFD tools used in nuclear reactor safety (NRS) and demonstrates the importance of this workshop as an excellent place for technical exchanges between experimentalists and CFD specialists.
- Regarding the numerical studies presented, it is worth noting the high number of presentations involving **open source software** (OpenFOAM, Code_Saturne, Basilisk, TrioCFD, etc.). OpenFOAM is currently used as the main tool in a collaborative initiative to build an open platform for the simulation of nuclear applications that was presented during the workshop. This is important for safety analysis since it could facilitate access to the numerical tools to perform studies or assess their validity. It is recommended that this trend be promoted.
- The topic of **uncertainty quantification** (UQ) in the CFD simulations was the main focus of three contributions: this is only a slight increase in the number of contributions with respect to the previous editions of CFD4NRS, although it clearly identifies UQ as a key topic for the application of CFD in nuclear safety assessment studies. The recommendation is to continue to encourage UQ methodology development for CFD and their application for NRS problems. This has been proposed as an upcoming WGAMA CFD Task Group activity.
- A session dedicated to **general frameworks** provided participants with a global overview of the experience of the French TSO (Technical Safety Organisation) when dealing with the use of CFD tools for nuclear safety studies and, from an industrial perspective, the CFD methodology developed by Framatome for fuel design. Such papers reveal a general tendency to develop the use of CFD tools for real applications in the frame of nuclear safety.
- The general feedback from the participants was that the workshop was well organised as a whole and the presentations were of good quality. At the end of the conference, several participants asked if the slides presented would be made available; approximately half of the slides have been shared with the participants. This event was successful in gathering a community to discuss and exchange ideas about recent activities in the development and application of CFD.

- In addition, the workshop hosted a side event to discuss the advances of the benchmark regarding the numerical study of the fuel structure interaction experiments of OKBM, involving several research teams worldwide.
- A virtual workshop did not allow for the benefits of an in-person meeting, such as networking, informal technical discussions (e.g. during breaks), or technical tours to local experimental facilities. The recommendation is therefore to organise the next workshop as an in-person event.
- High-quality communications from the workshop are being selected to appear in a special issue of the journal *Nuclear Engineering and Design*.
- The place and dates of the next CFD4NRS edition were announced at the end of the conference, and it was planned to take place at Texas A&M University, with Professor Y. Hassan as general chair, in 2022. The event was then postponed to 21-23 February 2023. During this workshop, all experts will hopefully be gathered in person to enhance technical exchanges, particularly on the following items: open source initiatives in the NRS, innovative CFD development and validation experiments, and maturity for fluid structure interaction simulation.

List of abbreviations and acronyms

AMR	Adaptive mesh refinement
ANS	American Nuclear Society
ATH	Advanced thermal-hydraulics
BPG	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
CHF	Critical heat flux
CLSVOF	Coupled level-set and volume of fluid method
CSD	Computational structural dynamics
CSNI	Committee for the Safety of Nuclear Installations (NEA)
DES	Detached eddy simulation
DNB	Departure from nucleate boiling
DNS	Direct numerical simulation
EWT	Enhanced wall treatment
FDS	Fire dynamics simulator
FSI	Fluid structure interaction
GRS	Gesellschaft für Anlagen- und Reaktorsicherheit (Germany)
GTFC	Goulier Turbulent Flame Closure
HZDR	Helmholtz-Zentrum Dresden-Rossendorf (Germany)
IAPWS	International Association for the Properties of Water and Steam
IPRESCA	Integration of Pool scrubbing Research to Enhance Source-term CAlculations
IRSN	Institut de Radioprotection et de Sûreté Nucléaire (French Institute for Radiological Protection and Nuclear Safety)
ITER	International Thermonuclear Experimental Reactor
JAEA	Japan Atomic Energy Agency
JHR	Jules Horowitz Reactor (France)
KAERI	Korea Atomic Energy Research Institute
KIT	Karlsruhe Institute of Technology (Germany)
KMOU	Korea Maritime & Ocean University
LES	Large eddy simulation

LWR	Light water reactor
MIT	Massachusetts Institute of Technology
MRV	Magnetic resonance velocimetrying
MSFR	Molten salt fast reactor
NEA	Nuclear Energy Agency
NRG	Nuclear Research and Consultancy Group (Netherlands)
NRS	Nuclear reactor safety
NSD	Nucleation site density
NURETH	Nuclear reactor thermal-hydraulics
OECD	Organisation for Economic Co-operation and Development
ONCORE	Open source Nuclear Codes for Reactor Analysis
PAR	Passive auto-catalytic recombiners
PIV	Particle image velocimetry
PTS	Pressurised thermal shock
PWR	Pressurised water reactor
RANS	Reynolds-averaged Navier-Stokes
RPV	Reactor pressure vessel
UQ	Uncertainty quantification
URANS	Unsteady Reynolds-averaged Navier-Stokes
SBES	Stress blended eddy simulation
SGS	Sub-grid scale
THAI	Thermal-hydraulics, Hydrogen, Aerosol, Iodine facility
TSO	Technical safety organisation
V&V	Verification and validation
VOF	Volume of fluid
VVER	Water-water energetic reactor
WALE	Wall-adapting locale eddy-viscosity
WGAMA	Working Group on Analysis and Management of Accidents (NEA)
ZLES	Zonal large eddy simulation

1. Workshop objectives and organisation

1.1. Scope and history of the CFD4NRS workshop series

The CFD4NRS workshop series of the NEA CSNI started in 2006 in Garching, Germany. It stemmed from the development of a new class of three-dimensional, fine-grained simulation tools: computational fluid dynamics (CFD) codes that gave access to more local structures of the flow than the classical, 0D-1D system-class simulation software usually applied in the nuclear safety field. However, to be applied to nuclear safety problems with a sufficient degree of confidence, CFD tools must be thoroughly validated against well-defined experimental data, and robust methodologies must be defined to tackle the most complex flows (potentially multi-phase) that can be encountered in nuclear reactors.

Consequently, the biennial CFD4NRS workshops aim to be forums during which numerical analysts and experimentalists can exchange new information on the application of CFD to nuclear power plants' safety and design issues. Both new experimental data for CFD code validation, as well as state-of-the-art (single or multi-phase) CFD applications are presented, with an emphasis on:

- **Experiments aiming to provide CFD-level validation data**, which include local measurements (multi-sensor probes, laser-based techniques, etc.). This kind of communication should include detailed information on initial and boundary conditions, mandatory for the subsequent CFD simulations, as well as a discussion of measurement uncertainties and error bound.
- **Single-phase or multi-phase CFD simulations** with a significant focus on verification and validation in connection with nuclear safety issues, which can include critical heat flux (CHF), pressurised thermal shock (PTS), pool heat exchangers, passive systems design, advanced reactor design, boron dilution, thermal striping and fatigue, and containment flows. The respect of the current best practice guidelines[1] for this kind of simulation is scrutinised in the review process. Discussion of uncertainty quantification (UQ) is also highly encouraged.

EDF R&D applied to be the organiser of the 8th edition of the CFD4NRS series in the aftermath of the 2017 edition of the NURETH conference; and associated with IRSN (the official French representative at CSNI) to formalise this candidature. EDF R&D has approximately 2 000 researchers and nine research centres (the EDF Labs) located in Europe, Asia and the United States, with an annual budget of over EUR 500 million. EDF R&D has carried out research in nuclear thermal-hydraulics for several decades, with a joint approach coupling experimental programmes and the development of dedicated simulation software.

Initially, the CFD4NRS-8 conference was scheduled for the beginning of September 2020. However, following another successful application, EDF R&D was also selected to organise the 2020 edition of the ANS ATH conference scheduled to take place at the beginning of April 2020. The dates were then shifted to late November 2020 to facilitate the internal organisation for EDF[2]. Postponing the event to 2021 was also a studied option,

[1] For instance, NEA (2015), "Best Practice Guidelines for the Use of CFD in Nuclear Reactor Safety Applications – Revision", NEA/CSNI/R(2014)11, OECD Publishing, Paris, www.oecd-nea.org/jcms/pl_19548.

[2] The ATH conference was eventually cancelled.

but finally not selected because of the planned organisation of the NURETH 2021 conference in Belgium[3].

With the COVID-19 outbreak, the possibility to cancel, postpone or organise the event online was discussed during the June 2020 meeting of the WGAMA CFD Task Group. The decision was made to make CFD4NRS-8 a virtual conference, changing its name to V-CFD4NRS8 (for "Virtual") due at once to the lack of clear foreseen worldwide improvement of the sanitary situation, the interest to have an opportunity to discuss the latest advances in the field in 2020 and the number of successful online events that had been organised at that time.

1.2. Main figures of the 8th edition

- There were 87 abstracts received from 18 countries, with one withdrawn by the authors (out of the scope of the conference). This represents a significant increase with respect to the previous edition, for which 76 abstracts had been received.
- Fifty full papers were received. The contributors who did not submit full papers mostly cited the impact of COVID-19 restrictions, such as a lack of access to laboratories or computation facilities.
- Approximately 110 participants attended from 14 countries: 6 from China; 45 from France; 15 from Germany; 7 from Korea; 7 from Spain; 6 from Sweden; 6 from Switzerland; 4 from the United States; 3 from Japan; 3 from the United Kingdom; 1 from Belgium; 1 from Lithuania and 1 from the Netherlands.

[3] Due to the COVID-19 outbreak, this conference was afterwards postponed to 2022.

2. Committees and acknowledgements

During the COVID-19 pandemic, the conference was made possible thanks to the efforts of a number of individuals, from local organisers to reviewers, in particular:

- Stephan Kelm (Jülich)
- William Benguigui and Pierre Moussou (EDF and IMSIA)
- Pierre Ruyer (IRSN) and Philippe Freydier (EDF)
- Didier Banner (EDF) and François Barré (IRSN)
- Nils Sandberg and Jeong Nam (formerly NEA), Martina Adorni (NEA)

Didier Banner was General chair of the committees and François Barré was General co-chair. The members of the scientific and organising committees are listed below:

Scientific Committee

- Henry Anglart (KTH)
- Emilio Baglietto (MIT)
- Dominique Bestion (Ind.)
- Sofiane Benhamadouche (EDF)
- Ulrich Bieder (CEA)
- Guillaume Bois (CEA)
- Abdel Dehbi (PSI)
- Philippe Freydier (EDF)
- Yassin Hassan (Texas A&M University)
- Thomas Höhne (HZDR)
- Stephan Kelm (Jülich)
- Jérôme M. Laviéville (EDF)
- Dirk Lucas (HZDR)
- Stéphane Mimouni (EDF)
- Wenxi Tian (XJTU)
- Fabio Moretti (NINE)
- Maria-Giovanna Rodio (CEA)
- Véronique Roig (IMFT)
- Pierre Ruyer (IRSN)
- Afaque Shams (NRG)

- Jan-Patrice Simoneau (EDF)
- Jinbiao Xiong (SJTU)

Organising Committee

- William Benguigui (EDF, IMSIA)
- André Bergeron (CEA)
- Guillaume Bois (CEA)
- Anthony Dyan (EDF)
- Olivier Fandeur (CEA)
- Mathieu Guingo (EDF) – co-ordinator;
- Nicolas Mérigoux (EDF)
- Pierre Moussou (EDF, IMSIA)
- Maria-Giovanna Rodio (CEA)

3. Technical contents of the workshop

The technical contents of the workshop were organised into four keynote sessions (consisting of invited lectures) and 16 topical sessions. In this section, the main points of the chairs' sessions are summarised.

3.1. Keynote sessions

Invited lecture 1
Tian Wenxi (XJTU)
CFD research activities in XJTU-NuTheL

Summary from the chair

Dr Tian Wenxi from the Xi'an JiaoTong University presented an overview of the numerous activities involving the use of CFD tools for nuclear applications at XJTU, focusing on some key activities, such as the use of cutting-edge open source or commercial tools to gain insights into the thermal-hydraulics of the reactor pressure vessel (RPV) and the fuel assemblies; the development of three-dimensional numerical tools to simulate the behaviour of steam generators and passive systems; and multi-physics (coupling between neutronics and thermal-hydraulics).

Invited lecture 2
Stéphane Mimouni (EDF)
Advances in physical modelling for multi-phase flows

Summary from the chair

Dr Stéphane Mimouni from EDF presented the latest advances in the physical modelling of multi-phase flows associated with the development of the in-house EDF simulation software NEPTUNE_CFD (which is based on a multifield Eulerian model), with an emphasis on the validation step. First, the components of the models implemented in NEPTUNE_CFD for the simulation of dispersed two-phase flows (bubbly flows, sprays) were described. Second, the main challenges and the key features of the model used for complex two-phase flows presenting large interfaces were shown. Third, the method currently under development at EDF to simulate flows in cracks was presented.

Invited lecture 3
Evgeny Shmelev (OKBM)
FSI benchmark to validate coupled CFD and CSD calculations

Summary from the chair

Evgeny Shmelev from OKBM presented the OECD fluid structure interaction (FSI) benchmark to validate coupled CFD and computational structural dynamics (CSD) calculations. His lecture paper presented the OKBM experimental setup, consisting of two in-line flexible cylinders in a water cross flow. He also presented the results of dynamic measurements of the flow and structure oscillations by different systems. More precisely, vibration acceleration amplitudes at the shedding frequency and at the cylinders' natural frequency were obtained as a function of the flow velocity. The velocity pulsation and pressure pulsation spectra were determined in the flow. A relatively simple structure enabled the use of non-contact measurement systems for cross validation and uncertainty quantifications. The influence on flow and structural reciprocal dynamics produced by the resonance between the vortex shedding frequency (harmonic) and the natural frequency was analysed. These activities were conducted to generate representative data in order to validate the requirements for hydrodynamic force calculation accuracy and to validate the methods of one-way and two-way coupled FSI calculations.

Invited lecture 4
Horst-Michael Prasser (ETH)
Film flow measurements applied to the annular flow in BWR fuel assemblies

Summary from the chair

Detailed knowledge on liquid films is important for different applications in nuclear safety research, but the most prominent is dryout in boiling water reactors (BWRs). Here, the interaction between the wavy film at the fuel rods and the droplets in the core flow determines the process. During the last decade, many investigations on film thickness measurement were done at ETH Zurich. A special liquid film thickness sensor was developed as a derivate of the wire-mesh sensor. It allows measurements of the dynamics of film thickness. The characteristics of the flow as wave velocities can be obtained directly. Using tracers, the average liquid velocity and the dispersion in axial and lateral direction can also be measured. In addition, information on droplet deposition can be extracted and pairs of sensors at opposite walls allow for combined film thickness and bubble shape and size measurements. The technique was used to investigate, among other things, the effect of spacers in BWR fuel rods. Specific dryout experiments were done in the dryout tomography experiment (DoToX), where the film measurement was combined with X-ray tomography. Here the influence of part-length rods was also investigated. In very recent investigations basing a Gauss-decomposition, the liquid fraction of the base film, ripples and disturbance waves can be distinguished. A lot of valuable data for CFD-code development have been validated and are available for use by the community.

3.2. Technical sessions

3.2.1. Session A1-morning: Multi-physics I

Hoene T. and S. Kliem (HZDR), “Detailed simulation of the nominal flow and temperature conditions in a pre-konvoi PWR using coupled CFD and neutron kinetics”

Papukchiev A. (GRS), “Numerical prediction of flow-induced vibrations in reactor relevant geometries”.

Wang Y. et al. (XJTU), “Numerical simulation of flow-induced vibration of steam generator U-shaped tube based on fluid-solid interaction method”.

Summary from the co-ordinator

This session included three presentations on the simulating process that couples thermal-hydraulics with another physics:

- The first one, given by Dr Thomas Höhne from HZDR, showed a detailed numerical study on an industrial configuration (pre-konvoi pressurised water reactor [PWR]) with the commercial code ANSYS CFX, which is coupled (off-line methodology) with a neutronic code to gain insight into the flows occurring in the reactor core (which is modelled with a porous medium approach). The calculation domain consisted in the RPV and parts of the hot and cold legs. A hybrid RANS/LES (SBES – stress blended eddy simulation) turbulence model was used.

The presentation covered the important features of the numerical model and the methodology used.

- Angel Papukchiev, from GRS, gave the second presentation, which dealt with FSI issues in reactor relevant geometries. It showed a comparison between an elementary validation test case (the Vattenfall experiment) and numerical results obtained by the commercial code ANSYS CFX-MOR. A hybrid RANS/LES – zonal large eddy simulation (ZLES) turbulence model was used, and the results obtained showed a generally satisfactory agreement with the experimental data.
- Yingjie Wang (XJTU) gave the last presentation of the session: a numerical study of the flow-induced vibrations in the U-shaped region of the steam generator. It was performed with the commercial code FLUENT with a detached eddy simulation (DES) turbulent model and a moving grid, suggesting this method could yield valuable information on the thermal-hydraulics of this part of the reactor.

3.2.2. Session B1-morning: Severe accidents

Bang K.-H. et al. (KMOU), "CFD simulation of jet break-up in fuel-coolant interactions in LWR severe accidents: Boundary layer stripping".

Sato F. et al. (MRI), "Development of CFD-code for severe accident scenario in Japan".

Whang S. et al. (POSTECH), "Comparison of the LES sub-grid-scale models for the simulation of the turbulent natural convective flow in-vessel molten pool".

Summary from the chair

In this session dedicated to the application of CFD methodologies to the modelling of various aspects of severe accidents, three presentations were given:

- The first, given by Prof. Bang of the KMOU, investigated the complex phenomenon of jet break-up in the context of fuel-coolant interactions with the commercial code FLUENT. A volume of fluid (VOF) model was used and the mesh influence was also investigated. The comparison with experimental data was performed with the COLDJET test case; the study suggests that the large-size droplets are produced by boundary layer stripping, and small-size droplets as an effect of Kelvin-Helmholtz instabilities.
- Dr Sato from MRI presented on the efforts made in Japan (MRI, JAEA, MHI) to develop an in-house simulation software based on OpenFOAM that is dedicated to severe accidents, including detonation. A collection of validation test cases was used (THAI, ENACCEF, PANDA, RUT), and developments to transfer data from FLUENT to OpenFOAM were also carried out.
- The work from Whang and Park presented an interesting inter-comparison of various sub-grid models for LES in the context of accurately reproducing the turbulent natural convection flows occurring in an in-vessel molten pool. A more thorough validation stage was presented as the next step of the work.

3.2.3. Session A1-afternoon: Advanced turbulence modelling I

- Abe, S. et al. (JAEA), "LES-WALE simulation on two liquid mixing in the horizontal legs and downcomer; the open test condition in the TAMU-CFD benchmark (IBE-5)".
- Camy, R. et al. (EDF), "LES calculations for the cold mixing benchmark from OECD".

- Dovizio, D. et al. (NRG), “Design of a DNS for a simplified PTS case with buoyancy effects”.

Summary from the chair

The session A1-afternoon: “Advanced turbulence modelling I” was led by T. Höhne (HZDR, Germany) and J.-F. Wald (EDF, France). This session had three presentations on flow mixing analyses. Two presentations were on the cold mixing benchmark IBE-5, while one was on the direct numerical simulation (DNS) for a simplified PTS test case.

In the first presentation, the author (S. Abe, Japan) presented CFD results on the open test condition in IBE-5. They selected the large eddy simulation (LES) solving the filtered equation of flow and concentration fields. Regarding the eddy-viscosity to model the turbulence flux of the momentum at sub-grid scale (SGS), wall-adapting locale eddy-viscosity (WALE) model, a modified version from the Smagorinsky model was applied. The comparison with the TAMU experimental data revealed the better performance of the WALE model.

In the second presentation, R. Camy (EDF, France) showed CFD calculations of the same benchmark IBE-5 using the CFD solver Code_Saturne. The methodology relied on a preliminary CFD calculation with an unsteady Reynolds-averaged Navier-Stokes (URANS) model, SST, then on more complex simulations with LES models. Overall, the agreement with experimental measurements was good. In the blind test, compared to other participants and the quantities exploited so far by TAMU, the results obtained by the authors ranked well. It can be noted that results were obtained from open source tools only and the calculation cost of a LES with 128M cells was limited to 24h on 1 680 cores.

In the third presentation, the D. Dovizio (NRG) presented a preliminary under-resolved DNS calculation on a simplified PTS scenario with the effects of buoyancy included. In particular, the Boussinesq approximation was used, while fluid properties were assumed to be constant. Cold water was injected and was consequently interacting with the wall of the downcomer, which was at a higher temperature. The aim of this work was to design a numerical experiment to generate a high-quality reference DNS database.

In this session, there was a limited use of best practice guidelines (BPGs) in the applications. Nevertheless, the session provided information of noteworthy scientific value regarding mixing processes to advance experiments and CFD technology and experience.

3.2.4. Session B1-afternoon: Multi-phase I

- Chen, T. et al. (EDF), “Numerical simulation of wall condenser and spray with neptune_CFD based on steam condensation experiments in the integral test facility”.
- Frederix, E.M.A. et al. (NRG), “Four-field large interface simulation of coexisting two-phase flow regimes”.
- Davy, G. et al. (EDF), “CFD modelling of two-phase flows in cracks”.

Comments by the chair

Tian Chen presented experimental results obtained in the enclosure COTHYD, which is dedicated to containment applications. The objective was to consider the interaction between a spray and a steam-air mixture. Measurements were compared with calculations performed with the NEPTUNE_CFD code, the multifield solver of Code_Saturne. A fair agreement was reached for the 2D cases, but unphysical results were shown in the 3D cases. The sensitivity to the turbulence modelling was assessed. The chair’s view is that only a k-ε model should be considered in this case because the mesh is too coarse, and there is a need for mesh sensitivity studies.

Eddo Frederix presented calculations performed with the OpenFOAM code. The objective was to develop a four-field model to adapt the closure laws specifically to bubbly flows, droplet flows and large interfaces. Rising bubble cases were shown. Four mass, momentum and energy balances were solved for each field. Perhaps a specific drag force should be added to couple continuous liquid and continuous gas fields.

Germain Davy presented calculations performed with the NEPTUNE_CFD code for the simulation of two-phase flows in cracks where the phase change is considered. In the simulations, four fields were considered: dispersed bubbles, dispersed droplets, continuous liquid and continuous gas. Two strategies were proposed resulting in the calculations and results corresponding. It is worth noting that although the overall results reached by these two strategies were similar, a supplementary regime field was observed when solving balance equations for each field. This conclusion should be investigated in other two-phase flows as boiling flows.

3.3. Conclusion

As recently as the 2010s, most modelling improvements were limited to the standard two-phase flow models because of computational limitations. Closure laws under algebraic formulation were targeted. As for 2022, supplementary equations have been solved, and the modelling of each field with accurate closure laws for interfacial transfer terms should be taken into account. This significant change in the modelling strategy demands even more robust and accurate numerical solvers. It could become difficult to distinguish physical effects from numerical artefacts in future calculations because of the complexity of the simulations.

3.4. Technical sessions continued

3.4.1. Session A1-evening: Advanced turbulence modelling II

- Evrim, C. and E. Laurien (IKE), “Large eddy simulation on thermal mixing of flows in T-junction geometries”.
- Howard, R.J.A. (EDF), “Thermohydraulic simulations of dead LEG flows”.
- Feng, J. et al. (MIT), “Assessing the applicability of the structure-based turbulence resolution approach to nuclear safety related issues”.
- Fabian, W. et al. (KIT), “Large eddy simulation of 5x5 rod bundle with split type mixing vanes”.

Summary from the co-ordinator

The four presentations were about the applicability of turbulent models aiming to be more precise (and solve more information) than the classical RANS approach, which consists of either LES models (from the presentation of Dr Evrim, Dr Howard and Dr Wiltshko), or the Massachusetts Institute of Technology (MIT) in-house hybrid RANS/LES STRUCT model. The general tendency seems to be that the RANS approaches remain – and may remain for years to come – the most widespread class of turbulence models used in the nuclear safety field. However, LES and hybrid approaches are becoming increasingly popular (especially for tackling highly unsteady flows), despite some challenges that need to be overcome (particularly related to the validation stage and computational cost). Hybrid approaches such as the MIT STRUCT seem to present an interesting compromise between accuracy and computational cost for mixing flows.

3.4.2. Session B1-evening: Multi-phase II

- Savinovskii, A.S. (OKBM), “CFD simulation of the DNB in tube and seven-rod bundle under VVER conditions”.
- Grazevicius, A. and Al. Kaliatka (LEI), “Numerical investigation of natural convection and thermal stratification phenomena in a rectangular enclosure”.
- Liao, Y. and D. Lucas (HZDR), “Simulation of bubble dynamics under pool scrubbing conditions”.
- Fan, W. and H. Anglart (KTH), “On verification and validation of turbulent vof simulations”.

Summary from the chair

Savinovskii, A.S. (OKBM), “CFD simulation of the DNB in tube and seven-rod bundle under VVER conditions”.

The authors presented a numerical study using the Euler-Euler model for boiling flow with the classical heat flux partitioning model and Peisman criteria on the alpha wall for departure from nucleate boiling (DNB). The nucleation site density (NSD) and bubble size at departure are based on validation test cases such as Bartolomei’s. The authors use data gathered by the critical heat flux tests with seven rods JSC experiment with a fuel assembly of the water-water energetic reactor (VVER) design. The problem of mesh sensitivity was discussed, but the approach indicates some accurate trends with regards to critical heat flux (CHF) prediction.

Grazevicius, A. and Al. Kaliatka (LEI), “Numerical investigation of natural convection and thermal stratification phenomena in a rectangular enclosure”.

The authors proposed the computation of a passive system-related configuration based on an experiment conducted by the Korea Atomic Energy Research Institute (KAERI) of a slander tank with a rather large vertical extension and exchanger in the bottom part where boiling occurs. Natural convection enhanced by bubbly flow is thus solved using k-ε realisable with an enhanced wall treatment (EWT) turbulence model. The results related to temperature prediction in the upper part of the tank were good, but there remained an issue with the erosion of the stratified layer at the bottom, which was not captured. Issues raised in the questions and answers concerned modelling for the interface at the free surface and the importance of dealing with lateral heat losses, including the necessity of meshing the walls for thermal resolution.

Liao, Y. and D. Lucas (HZDR), “Simulation of bubble dynamics under pool scrubbing conditions”.

This work is part of a large project within the IPRESCA (Integration of Pool scrubbing Research to Enhance Source-term CAlculations) consortium. It concerns the CFD simulation of bubble departure, fragmentation, swarm and break-up at the surface with a VOF approach in OpenFOAM. Two benchmark cases were considered. A simple case that enables a fixing of some mesh convergence issues and a more complex case. The simple case required validation against experiments with approximately 20 cells/diameter, whereas the second case had fragmentation within the swarm, which lead to more tiny bubbles (and remained unresolved). The questions and answers also showed how the meshing was still too coarse to catch coalescence.

Fan, W. and H. Anglart (KTH), "On verification and validation of turbulent volume of fluid simulations".

The work is based on the computation of co- and countercurrent two-phase flows in mainly two configurations (air-water separated by baffle and annular flow). The model used is the VOF model, and the authors presented first a verification step and then a validation step. A phenomenological correction approach enabled the performance of some fitting according to the damping coefficient of the turbulent model term in the k-ω SST model. This occurrence showed significant improvement in the results without having sufficient data to justify the modification. The possible use of DNS databases was suggested. The complexity of the problem of wavy liquid film with potential droplets entrainment was also discussed.

3.4.3. Session A2-morning: Design and plant application

- Brunet, D. et al. (TechnicAtome), "Validated CFD calculation tool for the Jules Horowitz Reactor (JHR) reflector hydraulic design".
- Catalán, D. et al. (IDOM), "Thermal-hydraulic code analysis benchmark of ITER vacuum vessel components by use of different CFD codes".
- Kang, H S. et al. (KAERI), "CFD analysis for a natural circulation flow between a reactor and a steam generator in the OPR1000".

Summary from the chair

The session included three papers dedicated to three highly different types of reactors: the research reactor, the International Thermonuclear Experimental Reactor (ITER), and the PWR.

The first paper concerned the French research reactor, which is called Jules Horowitz, and considered the design of its reflector part's cooling circuits. The full design process was described, including CFD calculations and corresponding experiments. The CFD commercial code StarCCM+ was used.

The second paper described a benchmark that was performed on part of the vacuum vessel of ITER, the fusion reactor. Three codes that originate from the main codes used within the CFD community were benchmarked: Star-CCM+, FLUENT and OpenFOAM.

The last paper related to an accidental configuration on the optimised Korean reactor OPR1000. The CFD-code ANSYS CFX models the steam generator, including an equivalent bundle, and was used to provide a system code with data. Different modelling was used.

All papers highlighted the usefulness of CFD and the convenience of such approaches, which have been well developed. The papers focused on the importance of validation and confidence. This confidence in CFD results was obtained by different means: experiments, benchmarks and sensitivity calculations.

3.4.4. Session B2-morning: Experimental studies I

- Bruschewski, M. et al. (UNI-ROSTOCK), "Full-field mean velocity and reynolds stress measurements in fuel assembly models using magnetic resonance velocimetry".
- In W.K., et al. (KAERI), "Reflood quenching experiment for cr-alloy coated accident-tolerant cladding"

- Yang, Y. and J. Xiong (SJTU), "Experimental measurement of air-water flow in narrow channel based on wire-mesh sensor".

Summary from the chair

This session was composed of three talks centred on innovative measurements or measurement methods.

The first talk, performed by Ms Kristine John, PhD candidate at the Institute of Fluid Mechanics of Rostock, presented a relatively new measurement method, the magnetic resonance velocimetry (MRV). This technique is an interesting addition to existing techniques because it provides the local time-averaged fields of velocity, temperature, particle velocity, void fraction, Reynolds stress tensor and turbulent heat flux in a non-intrusive manner and within very reasonable times (between minutes and hours). The spatial resolution of MRV is rather fine at about 0.4 mm and therefore the data provided by this technique is overall a very good candidate for making comparisons with CFD results. However, some limitations exist that are mainly related to the choice of materials, size of the rigs and temporal resolution.

There was no second talk because the speaker, Dr In Wang-Kee from the Korea Atomic Energy Research Institute, unfortunately could not join the conference in real time.

The third and final talk was presented by Mr Yiang Yang, PhD candidate at the Shanghai Jiao Tong University. The talk explored the influence of narrow channel walls on a bubbly air-water flow using a non-invasive variation of the wire-mesh sensor, which is called the wire-mesh photography sensor. The bubbles' rising velocity was obtained by this technique and compared with predictions obtained from models of a bubble rising in an infinite domain. Discrepancies of up to 60% were observed and attributed to the fact that the liquid velocity profile is modified in the presence of narrow walls, which is a neglected effect within the infinite domain assumption. A model that could provide corrective force was proposed and successfully reduced the deviation of the measurements by up to 6%.

3.4.5. Session A2-afternoon: Multi-physics II

- Bolshukhin, M. et al. (OKBM), "Adaptation and validation of CFD-code coupling with one-dimensional thermal-hydraulic code to calculate reactor plants taking into account two-phase flows for analysis of emergency operation modes".
- Dovizio, D. et al. (NRG), "Assessment of different combustion models in hydrogen safety management".
- Halouane, Y. et al. (LEMI), "Simulations of upward hydrogen flame propagation in the ENACCEF2 facility".

Summary from the chair

Dr Romanov from OKBM presented on the "Adaptation and validation of CFD-code coupling with one-dimensional thermal-hydraulic code to calculate reactor plants taking into account two-phase flows for analysis of emergency operation modes". The talk was about the successful coupling of a homogeneous CFD code (ANSYS CFX, 3D) with a homogeneous system code (TG1D, 1D). A first test case showed the consistency of the density field between the two codes, which both rely on the International Association for the Properties of Water and Steam (IAPWS) tables.

A second case modelling the ACME primary circuit (in the Chinese facility) enabled the comparison of the stand-alone TG1D code with the coupled model in a non-symmetrical situation. The benefit of the coupled model was proven in this case and further work will compare the numerical results against experimental data.

The second presentation was given by Dr Dovizio from NRG and was about the "Assessment of different combustion models in hydrogen safety management". Several combustion models for hydrogen combustion were assessed and validated on a slow deflagration in a closed and fan-stirred spherical vessel with controlled turbulence.

Better predictions of the Goulier Turbulent Flame Closure (GTFC) and GTFC+ models have been found. The fast deflagration to detonation (DDT) case was simulated with a density-based approach in ANSYS FLUENT, which has the different combustion models implemented through user functions. The results corresponded with the shock wave generation, reflection and interaction with the flame, although the GTFC predicted lower values for the flame speed.

The last presentation of the session was given by Dr Halouane from Laboratoire Energétique, Mécanique et Ingénieries (LEMI) Université M'hammed Bougara, Boumerdes, Algeria and was about "Simulations of upward hydrogen flame propagation in the ENACCEF2 facility". The talk concerned the results of the benchmark on hydrogen deflagration within the MITHYGENE project. Three cases conducted in the Enceinte d'Accéleration de Flamme (ENACCEF2) facility were simulated: the first one with open results, then two blind cases with only initial and boundary conditions known by the participants. Grids and time step sensitivities were tested on the open case within best practice guidelines.

The first two cases (11% and 13% hydrogen concentration) produced good predictions of the overall combustion process, and the interaction of the pressure waves and the flame was particularly satisfying. The third case (15% hydrogen) showed two limitations of the current modelling: the lack of an auto-ignition model and the fact that the turbulent flame speed correlation does not include the effect of the Lewis number.

3.4.6. Session A2-evening: Experimental studies II

- Freitag, M. and B. von Laufenberg (Becker Technologies), "Large field particle image velocimetry proving code validation data for containment CFD application".
- Veber, P. et al. (Ringhals AB), "Experimental and numerical analyses of cavitation in orifices plates for flow limitation in nuclear applications".
- Kossolapov, A. et al. (MIT), "Experimental investigation of bubble dynamics in sub-cooled flow boiling of water at prototypical pressure of boiling water reactors".

Summary from the chair

This session explored different experimental studies that are useful for the development and validation of CFD codes. Data obtained by particle image velocimetry (PIV) measurements in the THAI test vessel are essential for validation of containment CFD and specifically the generation and dissolution of helium/air stratification by natural convection. Particular attention was given to turbulence inside the Thermal-hydraulics, Hydrogen, Aerosol, Iodine (THAI) facility. The evaluation of the international TH-32 benchmark will complement this work.

Problems of cavitation induced by orifices plates were analysed experimentally and numerically. Different turbulence models were compared against each other to reveal their ability to predict cavitation. A method for predicting the risk of cavitation using CFD was described with the difficulty of properly determining minimum pressure. Simulations and comparisons with experimental data were planned for multi-holes and multi-hole orifices placed in series.

A new experimental facility to study the flow boiling at high pressure was presented. The facility will enable a better visualisation of the bubble and its footprint without interference pattern. The bubble departure mechanisms were observed to not be the same as at low pressure. These experimental data were promising and will improve the modelling of the boiling process with new boiling parameters such as bubble departure diameters and nucleation site density.

3.4.7. Session B2-evening: General frameworks

- Kelm, S. et al. (Juelich), "Recent progress in the international initiative on the development of an open source platform for E&T and R&D in nuclear applications".
- Rehm, M. et al. (Framatome), "Framatome's unified single-phase CFD methodology for fuel design and analysis".
- Ruyer, P. et al. (IRSN), "Using CFD in the frame of safety studies – some IRSN experiences".

Summary from the chair

Three papers were presented during this general session.

The first presentation by Stephan Kelm highlighted a collaborative work project and recent progress in the international initiative on the development of an open source platform used for education, training and research and development. Both the work and recent progress are connected to nuclear applications. The open source nuclear codes for reactor analysis (ONCORE) initiative was launched under the aegis of the International Atomic Energy Agency to promote the development and application of open source multi-physics simulation in support of ongoing research and education in nuclear science and technologies. Kelm presented the initiative's scope and objectives, summarised the status of some of the available open source software packages and outlined the first new collaborations that aim to extend the CFD package OpenFOAM.

The second presentation by Markus Rehm from Framatome was about a unified single-phase CFD methodology for fuel design and analysis. CFD has become an invaluable tool for the design and analysis of PWR and BWR nuclear fuel. Rehm showed that Framatome has developed CFD methodologies that are tailored to address specific problems related to nuclear fuel applications. Stand-alone methodologies have evolved over the years, reflecting improvements in physics modelling, meshing capabilities and new experimental data for validation. Improved computing resources led to the legacy modelling decisions based on trade-offs between model size and accuracy becoming obsolete. A unified modelling approach became not only desirable from a consistency point of view, but also as a cost-effective option. Consequently, Rehm showed that a comprehensive single-phase methodology, which relies on common meshing techniques and a turbulence model setup, has been established and optimised through a benchmarking process. Finally, Rehm highlighted some applications relevant to the development of advanced fuel assembly design, which are based on this unified methodology.

The last presentation was a shared experiment by the IRSN about the use of CFD in the frame of safety-related studies. Pierre Ruyer, Jérôme Roy and Clément Viron showed that the reliability of CFD results and computation capabilities have improved quickly over the last decades, which has led to an increasing number of industrial case uses for CFD. The presenters highlighted the growing usage of CFD in safety-related studies in particular, which is a relatively new but significant trend. This use suits the adaptation of both methodologies and assessment requirements. The IRSN, which is the technical safety organisation in support of the French regulator, considered the use of CFD from two perspectives. On the one hand, the IRSN has to analyse CFD results while assessing some safety studies provided by the utility. On the other hand, the IRSN uses its own CFD studies, either to perform comparative studies while assessing safety cases, or to explore other safety-related flow configurations. The three presenters shared IRSN experiences and perspectives concerning these specific uses of CFD. Finally, the presenters recalled the requirements of CFD and described some perspectives concerning its use in the context of nuclear safety studies.

3.4.8. Session A3-morning: Containment

- Kelm, S. et al. (FZ-Juelich), "Status of the tailored CFD solver 'containmentFOAM' for the analysis of hydrogen mixing and mitigation".
- Cammiade, Liam M.F. et al. (WSA, RWTH Aachen), "CFD modelling of wall condensation for mixed convection flows on inclined walls and design of experiments".
- Stewering, J. et al. (GRS), "Development and validation of a PAR model for OpenFOAM".
- Wang, X. et al. (KTH), "Pre-test simulation OF HYMERES-2 PANDA tests for steam injection into pool through load reduction ring holes".

Summary from the chair

The session featured four papers, three of which were related to transport processes in the containment atmosphere and hydrogen mitigation, and the fourth to thermal-hydraulics in a pressure suppression pool.

The first paper, presented by S. Kelm et al., summarised the "Status of the tailored CFD package containment FOAM for analysis of hydrogen mixing and mitigation". The OpenFOAM-based CFD package aims to represent containment pressurisation, atmosphere mixing, H_2 mitigation and aerosol transport. Two application-oriented validation cases were presented on gas mixing and mitigation as well as H_2 mitigation by means of passive auto-catalytic recombiners (PAR). The simulation results revealed a consistent representation of the experimental data and demonstrated general applicability to technical scale analysis. The discussion concerned concluding the strategy for the publication of the code and measures implemented to minimise potential user errors.

The second paper, presented by Cammiade, was entitled "CFD modelling of wall condensation for mixed convection flows on inclined walls and design of experiments" and discussed the effect of wall inclination on wall condensation rates. Two counteracting effects – namely the acceleration of the flow and the damping of turbulent transport due to buoyancy – were identified by shakedown experiments and scoping simulations with *containment*FOAM. The model was used to design an experiment for further experimental investigations into the SETCOM facility and corresponding model development.

The third paper, presented by Stewering, was entitled "Development and validation of a PAR model for OpenFOAM" and introduced an extended manufacturers' co-relation to simulate the recombination rates of a passive auto-catalytic recombiner. The model was implemented in OpenFOAM and validated against the THAI recombiner tests HR5 and

HR43 for counter current flow conditions. Simulation results revealed correlations with experimental data and previous implementation of the same model in the commercial tool ANSYS CFX.

The last paper, presented by Wang, was entitled “Pre-test simulation of HYMERES-2 PANDA tests for steam injection into pool through load reduction ring holes” and addressed the design of an experiment for a test in the PANDA facility within the NEA HYMERES-2 project. Special focus was given to the position and steam injection rates through the load reduction ring. The FLUENT simulations aimed to identify the occurrence of a stable stratification and sufficient transient duration for the conduction of detailed measurements of the flow field.

3.4.9. Session B3-morning: New reactors

- Bhatia, H. et al. (CEA), “Thermal-hydraulic analysis of the flow in the micas experimental facility using CFD”.
- Mimouni, S. and C. Peniguel (EDF), “Modelling of sodium boiling flows with neptune_CFD and application to the GR19 experiment”
- Pakholkov, V.V. et al. (OKBM), “Validation of the CFD model for the study of natural circulation in sodium-cooled fast reactors”
- Cartland Glover, G. et al. (STFC), “Molten chloride fast reactor draining transients”.

Summary from the chair

The session dedicated to a “new reactors’ design” consisted of four presentations. All four speakers presented work dedicated to the study of sodium or molten salt fast reactors (MSFRs). The work showed how CFD methodologies have largely improved regarding the study of these kinds of flows and correlation is evident when compared with experimental measurements. Moreover, the results show an efficient use of “simplified” hypotheses such as the “porous medium” model for complex geometries like reactor cores. Session A3-afternoon: uncertainty studies

- Lázaro, D. et al. (GIDAI), “Uncertainty analysis of FDS input parameters in fire simulations of nuclear power plants”.
- Ji, R. et al. (UBM), “uncertainty quantification for URANS based CFD analysis of buoyancy driven flows – comparison of the sensitivity of URANS and LES”.
- Acton, M. and E. Baglietto (MIT), “A data-centric method to quantify turbulence modelling uncertainty for nuclear reactor”.

Summary from the chair

This session addressed uncertainty studies, which is a topic of growing importance in CFD for nuclear reactor studies. The quantification of uncertainties has been used with existing methodologies for several years in studies using thermal-hydraulics system codes, but remains less developed in CFD. Nevertheless, people are now working fairly actively in this field.

The three presentations in this session covered different aspects of uncertainty studies, giving participants a wide overview of what is being done in this domain. The presentations covered input uncertainty propagation, epistemic uncertainties and methodologies for dealing with them.

A presentation by David Lázaro from GIDAI concerned the influence of input parameter uncertainties on fire simulation results using the fire dynamics simulator (FDS) code. This relates to important safety issues because an uncertainty about a parameter such as ventilation, the location of the fire or the thermal properties of cables can make a difference to the cable temperature and heat flux, which in turn can determined whether a fire occurs or not.

A presentation by Ruijun Ji from Universität der Bundeswehr München dealt with the uncertainty associated with a URANS turbulence model. This epistemic uncertainty was investigated through a comparison with a LES in a simple model. The LES was previously compared to a DNS, and produced similar results. The impact of using a URANS model was quantified on the simulation results (bias) as well as on the different propagation of input uncertainty (different sensitivity to input parameters) in a comparison with the LES model.

A presentation by Michael Acton (MIT) outlined an approach for quantifying model uncertainty in CFD: a turbulence model uncertainty which is associated with a turbulent viscosity model and quantified through a random field parametrisation approach.

3.4.10. Session B3-afternoon: Multi-phase III

- Okagaki, Y. et al. (JAEA), “Numerical study on bubble hydrodynamics with flow transition for pool scrubbing”.
- Frederix, E.M.A. et al. (NRG), “Towards direct numerical simulation of turbulent co-current taylor bubble flow”.
- Sayed, M.A. et al. (PSI), “Validation of WALL modelled large EDDY simulation against direct numerical simulation in particulate channel flow”.
- Mikuž, B. et al. (IJS), “Taylor bubble behaviour in turbulent flow regime”.

Summary from the chair

In this session about multi-phase flows, four presentations were given that focused on highly challenging high power computing simulations.

The first presentation, by Dr Okagaki (JAEA, Japan), dealt with bubble hydrodynamics in pool scrubbing conditions and focused on flow regime transition. The study was particularly focused on bubble sizes’ distribution and transition to oscillatory trajectories, which have important consequences for heat and mass transfer and aerosol removal efficiency. Simulations based on OpenFOAM (VOF or combined LS/VOF methods) were successfully compared with Abe et al.’s (2018) experiment with an injector. Both methods produced similar results, but the coupled level-set and volume of fluid (CLSVOF) method led to a slightly higher relative velocity.

The second talk, by Dr Frederix (NRG), focused on the high-fidelity simulation of turbulent co-current Taylor bubble flow. One challenge is to achieve an appropriate representation of turbulence behaviour at a large-scale two-phase interface and the subsequent break-up into smaller structures. A DNS of this configuration is under way to achieve accurate and converged predictions of the Taylor bubble skirt, and the related bubble shedding. RK-Basilisk simulations with adaptive mesh refinement (AMR) were performed and compared to previous LES with OpenFOAM. The loss of void of the Taylor bubble was much smaller than what OpenFOAM predicted. Nevertheless, there was still poor agreement with the experimental data of Shemer et al. because higher computational resources are required to achieve DNS.

The fourth presentation, by Dr Mikuž (JSI), addressed the same topic as the third, being based on the demanding wall-resolved LES VOF simulations in OpenFOAM. There were similar difficulties with capturing small bubbles produced at the trailing edge of the Taylor bubble, which causes a strong over-prediction and highly mesh-dependant estimation of the disintegration rate of the main bubble. Both approaches provided a sound basis for more general reduced order CFD models of all two-phase flow regimes.

Lastly, the third talk by Dr Sayed (PSI) focused on the validation of wall modelled LES against DNS in particulate channel flow. The good capabilities of the algebraic wall model LES were demonstrated by a comparison with DNS reference data at Re_τ=150 from Marchioli et al. (2007). The classical strongest particle clustering for mid-inertia Stokes numbers was successfully recovered. The goal is to apply this LES model to more complex 3D-flows.