SUMMARIES BY SESSION CHAIRMEN

Technical Session A1: Plant Applications (Chairman: M. Henriksson)

The papers presented demonstrated the power of CFD as a simulation tool, but gave some warnings too. The paper by Böttcher featured a detailed CFD model of a whole reactor vessel for a real plant coolant-mixing test. The degree of geometric detail was impressive. Significant too was the fact that the use of BPGs (use of 1st and 2nd order space discretisation schemes exposed deficiencies in the use of the two-equation model for turbulence.

The paper by Boros et al. addressed the issue of thermal stratification, both in a pressurizer surge line and in the surge line of a high-pressure injection (HPI) system. For the first application, CFD predictions were compared with plant temperature measurements, though additional information from the plant would probably have been helpful, and a sensitivity study for the pipe inlet velocity (at the pressurizer) could have been added. Nonetheless, the work demonstrated that CFD could be an effective method for investigating thermal stratification, or possible lack of stratification (HPI system).

The paper by Légrádi et al. showed how CFD could be used *a posteriori* to gain insight into an actual reactor plant accident (the incident at the PAKS NPP fuel storage vessel), and provide guidelines for prevention and post-accident clean-up. Deficiencies in the design were exposed, and a good description of the different stages in accident progression was presented. CFD calculations also gave a better understanding of the post-incident state inside the vessel.

Technical Session B1: Advanced Reactors (Chairman: J. Mahaffy)

This session's papers reflected the wide range of areas in which CFD can contribute to the design and safety analysis of advanced reactors. The Paper by Morii was the only paper at the workshop discussing validation of a CFD simulation including fluid-structure interaction, and looked at examined a new neutron reflector for the APWR. To minimize the chance of flow induced vibrations in this structure, a 1/5 scale test facility was created, and fluid structure calculations incorporating LES turbulence model were validated against data from this facility before being used to predict behaviour at full scale. Data from the experiment could be made available for a validation database, but there would be restrictions on geometrical information.

The paper by Johnson focussed on the flow in the lower plenum area of a new Very High Temperature Reactor (VHTR). The main issue here is the behaviour of materials at high temperatures, and the CFD calculations provide a more detailed description of the temperatures distribution. Since the layout of the core support columns resembled a cross-flow heat exchanger, the reported validation was based on a problem in the ERCOFTAC database. The paper showed the importance of transient modelling for this flow pattern, and discussed the application of the ASME guidelines associated with journal publication of CFD calculations. BPGs were applied, including use of a much reduced time step to eliminate time discretization errors, and a mesh convergence study.

In the paper by Kang et al.; thermal mixing in a pool driven by steam injection from a sparger was addressed. This was an interesting application of an analytic model driving boundary conditions to remove a very small two-phase region from the calculation domain, permitting application of single-phase CFD to the pool mixing problem. The approach was tested against transient temperature data from an experiment with steam injected into a large pool of subcooled water. There appeared to be some sensitivity to mesh resolution and it was not clear whether a mesh convergence was yet achieved. However, much more significant changes resulted from shifting from 1st to 2nd order upwind methods.

Technical Session A2: Benchmark Exercises (Chairman: P. Mühlbauer)

The two papers presented here were closely connected to recent EU projects: ECORA and FLOMIX-R, respectively. The paper by Andreani et al. reported results of blind simulations undertaken within ECORA of two buoyant jet experiments (without condensation) performed within the PANDA facility as part of the OECD/SETH test series. Eight participants were involved in the benchmarking exercise, using the FLUENT, CFX and TONUS codes. This benchmark exercise confirmed the importance of BPGs – when applied with expertise – even if the full set of recommendations cannot be applied for shortage of time, money and/or computing power. Application of even a limited set of BPGs led to surprisingly good accordance of results of measurements and simulations at the two selected time instants, even when the standard k-ε model of turbulence was used.

The paper by Toppila described simulations of PTS-related tests performed at the FORTUM facility for the VVER-440 reactor. The simulations extended those performed within the FLOMIX-R project. Commercial CFD code FLUENT 6.2.16 was mainly used. Heat transfer between fluid and solid structures was not considered. Again, the use of BPGs was in evidence here, with computations performed for two mesh densities (800K and 3800K cells) and two time steps used to demonstrate grid independence of results. Stratification in the cold leg and formation of a density-driven plume in the downcomer were reproduced in the simulation. It was stated that careful use of this CFD tool to thermal-hydraulic analysis of PTS of a real NPP is justified.

Technical Session B2: CANDU Reactors (Chairman: B. Smith)

The papers in this section dealt with safety issues arising in the operation of CANDU reactors. The paper by Kang reported a CFX-5.7 simulation of the CS28-1 LBLOCA/LOECC test performed by AECL. Novel features included in the simulation were hydrogen production from steam-zircaloy reaction and radiative heat transfer 50% of total heat transfer). Creditable was the fact that uncertainties in the test measurements were available, so the data are valuable for code validation. However, BPGs were not applied. Overall, CFD predicted the correct trends well, but there was some overprediction of temperatures. With more time and computer resources (number of meshes 4.3M), this would be a good example for applying BPGs in regard to sensitivity to parameters such as the absorption coefficient for thermal neutrons in CO_2 and the emissivity of the radiative surfaces.

The paper by Kim was concerned with simulation of experiment CS28-2 from the same series of AECL tests. Again, uncertainties in measured data were available. A distinguishing feature of this test was the eccentric placement of fuel rods in the bundle, so a full 360° CFD model was required for the simulation, leaving a rather coarse mesh configuration between rods. Nonetheless, the temperatures appeared to have been captured by the code (CFX-10), except for the pressure tube. In this case, radiation heat transfer was the dominant mechanism (87% of the total heat transfer), and it is assuring to see that the CFD code appears to handle it well. Again, the use of BPGs should be encouraged to try to identify sources of error, and this may be more important than moving to more complex situations involving transient conditions.

Technical Session A3: Novel Applications (Chairman: T. Morii)

The paper by Graf, a new model for the interfacial area transport equation based on the Rayleigh equation for momentum transport was proposed. One issue raised here is whether the dynamic change of interfacial area by bubble coalescence and bubble break-up can be adequately estimated by the Rayleigh equation of a single bubble. Nevertheless the approach is very interesting since it avoids the use of empirical parameters completely. The paper by Hassan described an LES (Large Eddy Simulation) calculation of gas flow between the spherical fuel elements of the HTGR (High Temperature Gas Rector), and how predictions compare with measurements of the instantaneous flow field obtained using the PIV (Particle Imaging

Velocimetry) technique. This study is interesting in that it describes the application of LES to a real flow situation in NRS. Both presentations supply no new experimental data suitable for CFD benchmark validation.

Technical Session B3: Containment Issues I (Chairman: M. Houkema)

Though it is generally argued that the use of CFD methods in containment modelling remains in its infancy, the papers presented in this section do illustrate the beginning of a reliable technology. The paper by Kljenak et al. described the implementation of a wall condensation model in the commercial CFD code CFX4.4 using a correlation based on bulk values. CFD is meant to give increasingly better results with successive grid refinement, but the opposite may be expected here. This problem was overcome using an appropriately tuned correction constant. Fairly good predictions were made with this model for the ISP-47 experiments TOSQAN and ThAI, but the question then arises as to which value to use for real-size reactor applications.

An overview of the achievements of the TONUS CFD code was presented in the paper by Kudriakov et al. The code has been specifically developed to model hydrogen release, dispersion, and combustion in containments. Validation examples were presented: for example, simulation of experiments in TOSQAN and combustion experiments in the RUTH facility. Active development is ongoing in the areas of turbulence and combustion modelling, further validation work is planned. The code is currently being used to model the EPR.

Technical Session A4: Boron Dilution (Chairman: T. Höhne)

This session was devoted to coolant mixing studies during boron dilution transients in PWRs. The mixing test facility ROCOM was described in the paper by Kliem et al., and proposals put forward for two different types of experiments to aid code validation. The first concerns the transport towards the reactor core of a slug of hot, under-borated condensate which has formed in the cold leg after a SBLOCA for a postulated natural circulation flow rate, and the second the propagation of ECC water in the test facility under natural circulation or even stagnant flow conditions.

The paper by Dury et al. dealt with CFD simulations of the Vattenfall 1/5-scale PWR model using the CFX code. The simulations were initially part of the FLOMIX-R EU 5th Framework Programme, but had been extended in order to examine mesh-dependent effects and to provide comparisons with predictions using FLUENT. The theme of CFX validation for boron dilution studies was continued in the paper by Graffard et al. with comparisons of code predictions against data from UPTF TRAM C3 test, followed by application to the 900 MW French PWR series.

Overall, the session demonstrated that there existed already comprehensive experimental data with high resolution in space and time for benchmarking CFD codes for boron dilution transients, with the prospect of more data in the near future. Further, CFD models were already being applied to real reactor situations. Nevertheless, the benchmarking activity did show in some cases that agreement between numerical and experiment data still has to be improved before the approach can be considered fully reliable.

Technical Session B4: Containment Issues II (Chairman: M. Heitsch)

In session B4 „Containment Issues II" three papers were presented. Two papers (presented by E. Porcheron and L. Blumenfeld) addressed containment spray experiments and data application. The third paper (M. Houkema) gave an overview on validation activities at NRG for CFX.

The paper by Porcheron et al. concentrated on experimental findings and data obtained from tests carried out at the TOSQAN facility. The tests included investigations on heat and mass transfer from droplets under cold and hot atmospheric conditions. Steam-air and helium-steam-air gas mixtures were used in the test series. Data will be made available through the Severe Accident Research Network of Excellence (SARNET) EU 6th Framework Programme. Though some doubts concerning the shielding of thermocouples against direct contact with drops and the reproducibility of test data, it was considered that with filtering, reliable data, suitable for CFD validation, can be obtained.

The paper by Malet et al. focused on code benchmarking against spray experiments at two scales, using data from the TOSQAN and the MISTRA facilities. The first part of the benchmark for TOSQAN was carried out blind, and second phase was open. The use of CFD codes was encouraged. A second benchmark, related to MISTRA tests, will be open at the beginning of 2007. Though the benchmarks are part of the SARNET activities, test data will also be more widely available for code validation.

The status of CFX validation exercises at NRG was presented in the paper by Houkema et al., with validation cases concentrating on gas mixing and condensation phenomena taken from a number of test facilities. Special models for bulk/wall condensation and mist formation were developed in-house for this purpose. The wall condensation model is described in more detail. Some discussion of the influence of turbulence models was also given. Of significance here was the recommendation of a combined use of lumped-parameter and CFD codes for containment best-estimate analyses.

Technical Session A5: Mixing in Primary Circuit (Chairman: U. Rohde)

The three papers in this session were concerned with different aspects of flow mixing in the primary circuit. The paper by Höhne et al. dealt with the boron-dilution event using data from the ROCOM facility and the code CFX as the CFD tool. BPGs were employed with respect to grid, time step and numerical discretisation, and different turbulence models were used. For the flow regimes investigated (low flow with density differences), Reynolds stress turbulence models were found to provide the best agreement with the measurements. Similar experiments at low flow buoyant conditions were proposed to be performed at ROCOM with advanced instrumentation to provide future data for CFD code benchmarking.

The paper by Chang et al. dealt with investigations on mixing at small scale. Measurements on the flow structure in rod bundles equipped with different types of grid spacers to enhance heat transfer were performed. The velocity field and turbulent energy distributions within the rod bundles (enlarged by a factor of about 3 in comparison with the real geometry) were obtained using Laser Doppler anemometry. Spacer grids without mixing devices and with split-type and swirl-type mixing vanes were investigated, and typical flow patterns for the different mixing devices identified. Strongly anisotropic turbulence characteristics were found. The measurement data are very valuable, because data on the flow structure caused by mixing devices in sub-channel geometry suitable for code validation are very limited. However, the data are proprietary but can in principle be made available based on mutual agreement.

The paper by Westin et al. concerned thermal mixing in a T-junction (cold water in a horizontal pipe mixed with hot water from a vertical pipe), and both experimental data and CFD simulations were reported. being an element of the primary circuit. For the CFD, different turbulence models (URANS, LES and DES) were applied. URANS calculations were not able to reproduce the experimental data, though the importance of secondary flows caused by an upstream bend was demonstrated. Calculations with LES gave results which agreed qualitatively with the experiments, but the grid used was too coarse for LES, and wall functions were employed. The temperature fluctuations obtained with DES were damped. Overall, it was demonstrated that LES is well suited for thermal mixing calculations with large-scale fluctuations, though in this case further grid refinement is indicated. Valuable data for CFD code validation can be provided to CFD users in exchange for results from their simulations.

Technical Session B5: Containment Issues III (Chairman: M. Andreani)

The session featured various CFD codes that have been validated for accident scenarios under a variety of assumed conditions, from those suited for analysis with lumped-parameter codes to more complex cases in which a 3-D predictive capability is required. Cases which fall into the latter (3-D needed) category include momentum-driven hydrogen distribution in a multi-compartment geometry (the paper by Wilkening et al.) – for example, simulation of the Battelle Model Containment (BMC) – and calculation of the effects of the hydrogen recombiners (the paper by Royl et al.). The use of CFD for studying detonation in 2-D was demonstrated against a number of validation tests in the paper by Redlinger.

Generally, it was shown that "old" data (e.g. including local inlet H_2 concentration at the inlet to the recombiners) could provide useful data for testing CFD codes. In fact, to obtain a full validation, velocity fields would be needed, but information on velocity is still too sparse, even for "new" data.

Technical Session A6: Stratification Issues (Chairman: D. Bestion)

The papers in this session were concerned with recent applications of two-phase CFD to stratified flows. The paper by Wintterle et al. described separate-effect tests carried out to investigate interfacial transfer and turbulence characteristics in air-water, counter-current flow, and associated modelling. Attention here was paid to the prediction of the average mean liquid velocity and turbulence intensity as well as the transverse void profile in the wavy free-surface region. The Kelvin-Helmholtz instability, and the transition from stratified to slug flow, was also investigated for an air-water, separate-effect test in the paper by Štrubelj et al., and simulated with CFD. In the paper by Vallée et al., Direct Contact Condensation at a free surface with condensation-driven instability was investigated for a horizontal pipe with subcooled water injection into steam.

In all cases, the two-fluid model, with a RANS approach to turbulence modelling using either the k-ω or k-ε models, was employed, and showed there were good prospects to simulate the free surface without any explicit interface tracking. Most of the modelling effort was paid to the interfacial transfers, and some advances were obtained. However, the modelling of the complex interactions between waves, turbulence, interfacial friction, and interfacial heat and mass transfers will require further investigation. These calculations also illustrated the difficulty in prescribing precise inlet and outlet boundary conditions for CFD simulations. A definite message here is that more attention should be paid in providing all the relevant experimental information to CFD tools when separate-effect tests are designed.

Technical Session B6 : Code Validation (Chairman: D. Lucas)

Two of the papers presented at this session were concerned with validation of CFD codes for two-phase flows, and one for single-phase flow. For modelling bubbly flows, it is important to consider the bubble size, since the interfacial transfers strongly depend on this parameter. This was the theme of the paper by Frank et al., in the context of the inhomogeneous N*M MUSIG model implemented in the CFX code. Validation of the model using data for the development of flow along a large vertical pipe was discussed.

Validation of a special customized module built on the foundation of the commercial CFD-code STAR-CD for BWR problems was the topic of the paper by Ustinenko et al. A very comprehensive validation strategy was introduced, based on adiabatic flow experiments (validation of interfacial drag and wall friction models), surface boiling experiments (validation of models for boiling, inter-phase heat and mass transfer, surface heat transfer, surface drag during boiling, boiling crisis), steam condensation experiments (validation of inter-phase heat and mass transfer models and interfacial drag models), dispersed flow of water droplets experiments (validation of models for inter-phase heat and mass transfer, droplet deposition on heated surfaces, and surface heat transfer), integral experiments in which several two-phase

flow regimes occur, and on model experiments in which two-phase flow in bundles of uniformly and non-uniformly heated fuel rods were studied.

The paper by Bieder et al. concerned validation of the TRIO_U code for boron mixing against data from scaled experiments, but included also a qualification procedure for full-scale nuclear reactor application. The objective of this dual procedure was to ensure that the validation calculations were performed with the same modelling hypotheses as the reactor analysis, for which no experimental data are actually available. A UPTF test and a buoyancy-driven ROCOM test were selected as the validation test cases, and BPGs were applied by introducing a stepwise improvement of the description of the main physical phenomena. Finally, simulation of the transport of a slug of low boron concentration in the primary circuit of a French PWR was presented.

Technical Session A7: Boiling Models (Chairman: F. Moretti)

The modelling of pool and convective boiling remains one of the great challenges for CFD, and the three papers presented in this session.

The paper by Yun et al. dealt with experimental investigations of so-called "downcomer boiling" phenomena occurring during the reflood period of a postulated LB-LOCA scenario in an APR1400 reactor. The test section was a rectangular heated channel, representing a down-scaled sector of the downcomer of the reactor, and included an inlet nozzle simulating a Direct Vessel Injection (DVI) device. Instrumentation allowed visual observation, as well as the measurement of local two-phase flow parameters (liquid and gas velocity, void fraction). The experimental investigation provided insight into the complex two-phase flow occurring in the downcomer, including bubble nucleation (on the inner side), recirculation and etc.

The validation procedures for the cavitation and boiling models implemented in the NEPTUNE-CFD code were described in the paper by Mimouni et al., with experimental data provided by Super Moby Dick, DEBORA and AGATE. Part of the work focused on the secondary flows and void distributions resulting from the presence of mixing devices or obstacles (such as spacer grid). This work represents one of the few attempts to address real rather than academic channel geometries.

Some computational studies on forced convective boiling flows in a heated pipe were recounted in the paper Končar et al., together with the related validation activity, which was based on data from the literature (from Arizona State University). The CFX-5 code was used for the numerical work, and attempts were made to follow BPGs during this exercise. Though reasonably good agreement between numerical and test data could be obtained with built-in models, some aspects requiring further modelling effort were identified, such as the prediction of the liquid velocity profile and the near-wall void profile.

Technical Session B7: Containment Issues IV (Chairman: J. Mahaffy)

The paper by Royl et al. described validation of GASFLOW II against HDR test E11.2 and ThAI tests TH10 and TH13. The work included a very basic mesh sensitivity study and comparison of results from first-order upwind and second-order Van Leer methods. By normal standards, the mesh was fairly coarse, but this resulted from practical run-time restrictions associated with the long transient. There is a continuing debate between these analysts, who claim that failure to predict atmospheric stratification in Phase III of TH13 was the result of incorrect specification of boundary conditions, and the ThAI experimentalists. An experiment with more instrumentation is needed to settle this dispute. Audience members noted that k-ε turbulence models always have problems near injectors, and that variation of the injection area in the simulation has a potential for cancellation between discretisation errors and those due to the turbulence model.

A new particle tracking model for the FLUENT code was described in the paper by Dehbi which was designed to improve prediction of particle deposition rates at walls by superimposing velocity profiles derived from near-wall DNS calculations. The model was validated using data from the bio-medical field for deposition of aerosols in an idealized human airway, where significant improvement over the standard model was demonstrated. In response to questions from the audience, the author confirmed that he considered that this approach could serve as a framework for particle re-suspension, and that he did not expect the use of Reynolds stress predictions of boundary velocities over those of DNS.