SUMMARIES BY SESSION CHAIRMEN

Session HOR: Horizontal Flow – Pipe Flow (Chairmen: Chul-Hwa Song and T. Watanabe)

In the Session HOR, three papers were purely experimental studies and two papers presented both experimental and numerical studies.

In the experiment by Marchand et al. (HOR-03), velocities and void fraction have been measured in a horizontal pipe with several flow regimes including bubbly and stratified flows in air-water conditions; Vallee et al. (HOR-05) also investigated steam-water counter-current flow to simulate a hot-leg model with steam generator inlet plenum. Area-averaged velocity and void fraction in air-water pipe flows have been measured using a nuclear magnetic resonance (NMR) method by Lemonnier (HOR-02), and eddy diffusivity and dispersion coefficient were obtained. All these experimental works were performed for producing experimental data, which will be used for the development and validation of CFD codes and models. Measurement uncertainties were not systematically determined but some effort was noticed to compare several measurement techniques when possible.

Numerical simulations using the NEPTUNE-CFD code have been performed for condensation in counter-current steam-water stratified flows by Strubelj and Tiselj (HOR-04), and also for slug formation in air-water stratified flows by Bartosiewicz et al. (HOR-01). Such stratified to slug flow regime transitions were shown to be simulated well.

The noticeable point indicating the current status in this session is that two simulations and two experiments were for the development of the NEPTUNE-CFD code, whereas the CFX code was mostly used for simulations of horizontal or stratified flows in the previous CFD4NRS in 2006. For development of two-phase CFD codes, modeling of the interfacial momentum and heat transfers including turbulent parameters near the interface are of importance and further studies are still necessary.

Although precise measurements have been conducted in the experiments, the importance and difficulties in the measurements of local and instantaneous values of variables near the interface were pointed out, and the size effects and the setting of the boundary condition were also discussed. These points should be carefully taken into consideration in the experiments.

It seems that the Best Practice Guideline (BPG) have been better considered in numerical simulations when compared with the previous workshop. It has been shown in this session that the flow transitions in horizontal flows could be simulated well for simple cases. Validations for more complicated flow situations including integral scale phenomena and bubbly/stratified/slug transitions would be desirable.

Session AC: Accident Analysis (Chairmen: N. Seiler and T. Morii)

This Session on Accident Analysis gathered various research subjects, such as insulation debris transport phenomena in water flow, Fluid Structure Interaction (FSI) related to the early phase LBLOCA, specific problems for severe accidents such as Steam Generator Tube Rupture, and molten corium coolability.

As these subjects are very different from each others, it is difficult to draw a general conclusion, but it appears that, with the progress of computer power, some CFD simulations of these accidents are now feasible.

Modeling used in the CFD codes seems to be mature enough to be applied to accident analysis issues and the presented works have shown some promising results. Simulations of a free surface and two phase flows configurations have been addressed in case of a recirculation sump blockage issue occurring either in Korean advanced nuclear power plant or in a PWR. Two ways to compute the two phase flows have been considered: a VOF method or an Euler/Euler approach. Considering the latter way, an important work has been fulfilled to validate the required modeling and also to adjust the model parameters. Other presented studies dealt with single phase flows simulated by commercial codes. The SGTR as well as the corium coolability problems have been addressed using the FLUENT code whereas, in the FSI problem, the thermalhydraulic part was simulated by the Star-CD code.

Regarding the corium coolability in a BWR, a new simplified model has been developed, called the Phase-change Effective Convectivity Model. Further improvements are still foreseen and in progress such as the accounting of entrained gaseous bubbles influencing the fiber sedimentation in case of insulation debris transport phenomena in water flow or the stratification of corium.

All presentations have included experimental validation and shown good agreement between calculation results and experimental data. A paper presented new data obtained in 5000 flow conditions of flashing, subcooled or saturated water, as well as critical flow data at pressures up to 9.3 MPa. The available data for codes validation are pressure differences and flow rate under single phase conditions, and critical flow rates as a function of inlet subcooling and pressure.

The BPG of the OECD GAMA seem to have been quite well followed and, simulation results have been validated against well instrumented and dedicated experiments. Some experimental tests have been used therefore to assess required unknown coefficients like drag force coefficient or other parameters. Furthermore, mesh quality has been checked. Meshing refinement and time discretization as well as sensitivity studies to numerical methods and to closure models like the turbulence models, have also been carried out as recommended by BPG. However, the larger and more complex geometry of a real plant would need a larger number of grid points than that used for these studies focused on particular regions of the NPP.

Presentations proposed in this session have also underlined, on the one hand, the necessity to perform multi-physic simulations involving codes coupling in particular in Fluid/Structure field. On the other hand, it has been shown that validated CFD simulation results could be used as input data to derive required further modeling like 1D deposition model for SGTR accident.

Session PTS: Pressurized Thermal Shock & Direct-Contact Condensation (Chairmen: M. Scheuerer and M. Andreani)

In this Session, 4 papers dealt with CFD analyses concerning applications to PTS in single and two-phase flows, to steam discharge in a pool, to air bubble entrainment caused by an impinging jet on the free surface, and to steam jet injected into a pool of sub-cooled water. Two paper presented experimental studies of a steam jet injected into a pool of sub-cooled water.

The paper PTS-01 presented a CFD modeling approach of a PTS two-phase scenario including the so-called Large Interface (LI) modeling, and the validation against air-water stratified flow data, and COSI steam-water condensation data. The NEPTUNE-CFD results obtained with the LI model were encouraging, the main trends being fairly well simulated, but further model improvement and validation require new experiments with more detailed measurements, in particular of the velocity field.

Simulations in paper PTS-02 with the FLUENT CFD code of an OECD ROSA test with temperature stratification in the cold legs and in the downcomer during Emergence Core Cooling System (ECCS) injection followed BPG for the mesh generation, and for the quantification of numerical errors. Sensitivity tests investigated the effect of turbulence model and spatial discretization schemes. A Reynolds Stress turbulence model provided the best results, and gave satisfactory agreement with data. Further improvements would require detailed velocity measurements.

The papers PTS-03 and PTS-04 present data and CFD analysis of steam jet condensation experiments with local velocities and temperatures measurements. Simulations with ANSYS CFX with a modified condensation model predicted well the turbulent jet region before the tank wall but failed to predict the flow at the upper and lower region after the jet collides with the wall. New experimental data investigate a free turbulent steam jet and pool mixing induced by the jet using non-intrusive optical Particle Image Velocimetry (PIV). The measuring techniques are given with an error analysis.

The simulation in paper PTS-05 of air entrainment caused by a jet impinging into a liquid pool is a difficult challenge for CFD since the corresponding physical process takes place at very small length scales. CFX11 simulations showed the influence of the drag model and of the algebraic interface area density model on the calculated gas entrainment.

The paper PTS-06 described an experiment of steam discharge into sub-cooled water with condensation. Simulations with NEPTUNE CFD using several ‘established’ condensation models, such as the Hughes & Duffey model, overestimate condensation rates by up to two orders of magnitude. The DNS-based model developed by Lakehal et al. (2008) yielded satisfactory agreement with data.

Overall the quality of the presentations in the PTS session was good, and there is progress which can be summarized as follows:

- There is visible progress in the area of PTS and DCC investigations. Useful experiments, especially for pool mixing and jet condensation have been reported. CFD modeling increasingly addresses two-phase flows with phase change. There are more applications of BPGs, and more separate validation of individual models and components. In the experiments the need to control boundary condition of the fluid domain has been recognized as an essential input for CFD calculations.
- However, the modeling of momentum, heat and mass transfer at the interface requires further work. Flows combining several interface configurations such as a bubble flow with a free surface are still poorly modeled. More general and/or specific local models need to be developed. Numerical robustness also seems to be an issue for some codes when applied to two-phase flow with condensation and evaporation.
- Available experiments still lack measurements of local velocity, turbulence, void fractions, bubble sizes and bubble distributions. Small scale/simplified experiments with better defined initial and boundary conditions are required for the CFD simulations. Although some measurement uncertainties were specified, systematic uncertainties are rarely given.
- Although BPG are increasingly applied many results are still presented on single-grids without convergence checks or other information about numerical errors. This makes model assessment very difficult and limits the usefulness of these models for industrial applications.

Session MIX: Mixing Issues (Chairmen: B. L. Smith and D. Lucas)

A common feature of the papers, and an improvement since the last Workshop, was the general awareness that the application of Best Practice Guidelines (BPGs) is now a base requirement for CFD modeling. However, for purely practical reasons in terms of CPU overheads, there has to be some compromise in demonstrating mesh independence of solutions, for example for the VVER-1000 primary loop model, but maximum convergence of residuals was generally observed, and some sensitivity studies in regard to input parameters and (RANS) turbulence models. In contrast, most of the measured data are still being presented without error bars.

With the upgrade in computer hardware, and to a lesser extent software too (parallelization), the CFD simulations are now becoming more adventurous, with attempts to model explicitly core bundles and those primary circuit components for which 3-D representation is needed. Calculations involving some tens of millions of meshes are now becoming more common, and being used to simulate transients too. However, meshes next to walls are still too large, even for the application of high Reynolds number turbulence models, and require further mesh refinement.

Several papers dealt with thermal mixing in T-junctions, and several sources of CFD-grade experimental data (meaning high resolution in space and time, instantaneous velocities and turbulence statistics) were identified. The need to employ advanced turbulence models (SAS, DES/LES) was noted, even for time-averaged data; RANS predictions were generally poor. In addition, for thermal fatigue studies, it is necessary to capture the dominant turbulence scales. It was again emphasized, however, that mesh dependency remains an intrinsic feature of LES/DES approaches in physical space, since mesh refinement changes also the filter width of the resolved field. Hence strict application of BPGs is not possible.

Innovative measuring techniques for producing the comprehensive local velocity and temperature (or passive scalar) data needed for CFD code validation were evidenced in several papers. For example, a 10 kHz planar array sensor has the possibility to measure non-intrusively film thicknesses in opaque fluids, and signals from wire-mesh sensors enable contours to be drawn over an entire pipe cross-section. These data complement those traditionally made using thermocouples.

CFD-grade data collection (over 4000 points) was reported in the downcomer section of the ROCOM test facility. There were also CFD-grade data collected in a vertical mixing test (VEMIX). A suspicion was advanced that outstanding discrepancies in boron dilution studies may be due more to the use of inappropriate turbulence models (to date RANS-based) than lack of geometric complexity in the models. The use of LES may be warranted, though if employed improvements need to be made in the near-wall treatment, the damping of maxima due to filtering, and a rationale for filter-width determination.

Session BOI: Boiling Flow, Bubbly Flow and critical heat flux (Chairmen: E. Hervieu and M. Henriksson)

In the session BOI, six papers presented modeling & CFD calculations, three papers presented measurement techniques, and two papers presented experimental data bases.

Not so many authors referred to BPGs, and they were only partly applied in some cases.

The main modeling issues dealt with bubbly flows and treated wall models, bubble dynamics, transport of interfacial area concentration, and turbulence modeling. A lack of validation data was pointed out for the wall heat transfer coefficient, the bubble departure diameter & frequency, the expressions of forces, and for the complex interactions between bubble dynamics and the turbulence. Source and sink terms for interfacial area concentration transport equations are still a major concern. K-ε models are used as « standard » but advanced models like RST are necessary to describe swirl.

A trend is noticed toward more complex geometries, which require mesh refinement and high CPU cost.

Measurement techniques tend to progress towards higher space and time resolution, toward complex geometries (pipes with internals) using non intrusive techniques, and toward representative thermal-hydraulics conditions.

Innovative work was presented with photon attenuation (X, γ), and miniaturized wall electrical sensors. Such advances require long-time development.

Presentations revealed that large efforts have been devoted to the assessment of measurement techniques by comparison with reference techniques, the achievement of dedicated calibration procedures, and rigorous accuracy quantification.

Targeted 3D experimental databases dedicated to CFD validation were presented using either air/water or simulation fluids to represent steam-water flows. Experimentalists devoted large efforts to the accuracy quantification concerning the measured data, the operating conditions, and the documentation that makes the data usable.

There still remain lacks in bulk velocity measurement, near-wall measurement, in the use of representative geometries, and in the investigation of nominal reactor thermal-hydraulics conditions.

Session MS: Multiscale Analysis (Chairmen: D. Bestion and Y Hassan)

This session, together with the lecture of Pr. Banerjee was a little different from the others. Here, CFD is not used directly for safety application but fine scale simulations are used for helping more macroscopic tools such as system codes or CFD with an averaged approach (RANS) to better model some specific small scale phenomena. Various techniques are used, such as DNS (Direct Numerical Simulation, LES (Large Eddy Simulation), with ITM (Interface Tracking Method), or ISS (Interface and Subgrid-Scale), which may bring precious information that available experimental techniques may not be able to provide.

Some progress has been obtained in the development and use of these two-phase fine scale approaches and the nuclear reactor safety community encourages research people to further develop and apply these tools focusing on some issues of interest. In the future, it might be proposed that people who develop closure relations for macroscopic approaches (system codes and CFD with an averaged approach) establish a list of small scale phenomena which are not well known, not measurable even with up to-date techniques, and which could be better known by such small scale numerical simulations. The nuclear safety community would give challenges to the scientific community for a better synergy between them and at the end for a better treatment of some safety issues.

Session CO: Containment thermal-Hydraulics (Chairmen: J. Mahaffy and E. Graffard)

In the session CO papers presented either experimental work or CFD applications to containment thermal-hydraulics, including mixing of steam-helium-air (helium being a substitute of Hydrogen), spray behavior and effects, jet flows, condensation and re-vaporization, and catalytic recombiners.

Some progress was observed since the last workshop:

- Gradual improvements are evident in two phase CFD tools such as NEPTUNE-CFD and GASFLOW
- Interesting new instrumentation concepts are reported in particular in the PANDA facility

- New numerical model are proposed for droplet evaporation at the wall

However problems are reported with liquid film modeling and condensing surfaces and questions are raised about modeling re-vaporization of those condensate liquid films. It was also underlined that diameter variations of the droplets caused by collision, fragmentation, and coalescence, should be taken into account in future calculations.

There is a lack of separate effects experiments about sprays for validation of CFD application to containment simulations. If existing experiments cannot be applied to this issue, new experiments are needed. There was also an issue concerning potential problems with the reliability of gas concentration by measuring the sound speed, when a fog of condensing steam is present.

All papers showed some level of mesh sensitivity study and some attempts to justify numerical choices

Not attempt to quantify uncertainties was presented.

A better idea of scale effects in containments is still required, which is: certainly a real challenge!

Session CSG: Core and Steam Generators (Chairmen: M. Andreani and F. Moretti)

The paper CSG-01 described an iterative process adopted at Westinghouse where CFD models are continuously improved and upgraded on the basis of new experimental information and increasing computational resources as well. Some experimental work and code assessment activity were also presented. The paper CSG-02 dealt with experimental investigations on flow mixing in the upper part of a VVER-440 coolant channel. In particular results from PIV and LIF measurement were shown, providing velocity and temperature distributions in the observed region and evidencing the effects of geometrical discontinuities on the flow field.

Papers CSG-03 and CSG-05 dealt with the CFD modeling of the SG of VVER reactors; the purpose of such activity was to obtain accurate information on the flow distribution over the SG tubes and thus help reducing the uncertainties connected with SOCRAT system code nodalization. The paper CSG-04 was related to the experimental and numerical investigation of pressure losses across spacer grids. The effect of the inclination of the mixing vanes was considered as an experimental parameter. LES was used in order to understand and clarify the main features of the turbulent flow downstream of the spacer, then RANS calculations were performed and results were compared against measured data. Although very interesting, the presentations seem to be not fully representative of the large amount of work that is being done in the related fields of Core and SG simulation, and may not reflect the state-of-the-art. Therefore also the present comments have to be taken with the due care since they are based on a limited set of results.

In comparison with the last Workshop, some progress has been observed:

- Detailed validation methodology, including stepwise approaches, are being developed for the calculation of heat transfer in bundles, including spacer effects.
- Better data are available which include velocity fields.
- Some details such as the azimuthal distribution of heat transfer parameters are now being addressed, and CFD-grade information of velocity fields are obtained.
- More and more attention is being paid to the estimation of experimental uncertainties and interpretation of the discrepancies between experimental data and CFD analyses.

All the works presented dealt with single phase flow and some difficulties related to turbulence modeling in complex geometries are reported. In relation to steam generator modeling, the large variety of accident conditions suggests to use various models for SG depending on the specific issue. There is probably a need to group such issues, for instance depending on water inventory on the secondary side.

A very interesting methodology was proposed to use data at low-Re number to compare qualitatively results obtained by CFD simulations for typical conditions (much higher Re number). However data in core conditions would be required for a proper code validation.

For the simulation of core and steam generator the current computer power basically prevent from applying existing BPG, because of the complex geometries: typically several million nodes are necessary (at least) for a detailed representation of the geometry and a refinement is basically impossible. Mesh refinement is possible only for special cases such as calculation of pressure losses across spacer grids.

Session AR: Advanced Reactors (Chairmen: T. Schulenberg and F. Ducros)

The session AR presented experiments and CFD simulations related to gas cooled reactors and to Sodium cooled reactors.

Measurement systems for heavy liquid metals are still in an early stage of development compared with the advanced non-invasive methods used with air or water. There is certainly a lot to be done in future to develop reliable measurement systems.

Gas entrainment and two phase flow modelling has been the subject of 3 papers about sodium cooled reactors. An interesting and novel approach for the numerical study of such problems has been introduced by Kei Ito et al., which was discussed in the session. They used a high precision volume of fluid algorithm which was formulated for an unstructured mesh. The method is volume conservative and phase velocities are defined independently using volume fraction values. Comparison with a single bubbly experiment was used for verification. This example shows that the simulation of a free surface flow can still be improved significantly by a more precise volume of fluid approach.

Another general subject of discussion was the importance of computer hardware configuration and system programming to optimize CFD for large grid applications. A closer interaction of computer system developers with CFD users was recommended to make best use of existing hardware.