Main results of the European project NURESIM on the CFD-modelling of two-phase Pressurized Thermal Shock (PTS)

The European Platform for Nuclear Reactor Simulations, (NURESIM project 2005–2008) addressed the creation of a Common European Standard Software Platform for modelling, recording, and recovering computer data for nuclear reactors simulations. One work package of the project was dedicated to the analysis and improvement of CFD capabilities for the simulation of two-phase PTS problems. Some SB-LOCA scenarios lead to a situation in which the cold leg is partially or totally uncovered when the Emergency Core Cooling injection is activated. The resulting complex two phase flow can be divided in characteristic flow regions: the jet flow with a free surface between steam and water, the zone of jet impingement, the horizontal two-phase flow and the flow in the downcomer. Many phenomena have to be reflected in a simulation of each separate region, but also when the simulations are coupled reflecting the integral process which is required to predict the thermal loads at the RPV wall. After analyzing the experimental database available for CFD model development and validation and identifying shortcomings of the models, different activities were dedicated to the simulation of single flow regions as well as the integral flow. Based on these experiences recommendations for the CFD-simulation of the two-phase PTS problem were obtained.


1 The two-phase PTS problem

Pressurized Thermal Shock (PTS) and Direct Contact Condensation (DCC) were identified by the European project EUROFASTNET as two of the most important industrial needs related to nuclear reactor safety where CFD may bring a real benefit [1]. The two issues are interrelated since DCC is a very important phenomenon in two-phase PTS scenarios. One typical PTS scenario limiting the Reactor Pressure Vessel (RPV) lifetime is cold water Emergency Core Cooling (ECC) injection into the cold leg during a hypothetical SB-LOCA (Small Break – Loss Of Coolant Accident). The injected water mixes with the hot fluid present in the cold leg and the mixture flows towards the downcomer where further mixing with the ambient hot fluid takes place. Such a scenario may lead to high thermal gradients in the structural components and consequently to thermal stresses. Therefore, the loads upon the RPV must be reliably assessed. The fluid at the location of the injection can either be in single-phase or in two-phase condition, depending on the leak size, its location, and on the operating conditions of the Nuclear Power Plant considered.

The NURESIM sub-project 2 (Thermohydraulics) Work Package 2.1 was focused on a two-phase flow configuration resulting from a partially or fully uncovered cold leg. In the case of a partially uncovered cold leg, a stratification of cold water on the bottom of the cold leg with counter-current flow of hot water and steam on top of this cold-water layer may occur (see Fig. 1). There is mixing between hot and cold water. Condensation takes place at the cooling water jet and at the free surfaces. Mixing and condensation are strongly dependent on the turbulence in the fluids. Finally, there is plume cooling in the downcomer. If the water level in the downcomer has dropped below the cold leg nozzle, cold water is injected into vapor and direct contact condensation takes place on the surface of the sub-cooled water. Stripe cooling will occur in the downcomer, leading to high thermal loads of the RPV, due to the fact that the RPV-wall is cooled by a cold water stripe of small width with a large temperature difference to the surrounding. DCC is of prime importance in this
situations since it is the main heat source for the cold water. It is strongly influenced by the interfacial structure and by the turbulence. Then interfacial transfer (momentum – including turbulence – mass and energy) have to be considered firstly in the jet area and secondly in the stratified flow.

As shown in Fig. 1, different flow phenomena occur. There are flows with separated surfaces (jet interface, horizontal interface), but also dispersed flows occur due to bubble entrainment (at jet impingement and possibly also in the horizontal flow region by entrainment caused by waves). Since there is a strong thermal non-equilibrium at these interfaces momentum transfer as well as heat and mass transfer have to be considered. The various two-phase phenomena taking place are strongly coupled with each other and also with the heat transfers at the system walls. The different phenomena depend on very different characteristic length scales, from the size of the smallest eddy up to the system scale. Some of the involved phenomena are not yet fully understood. Examples are the process of bubble entrainment and the influence of turbulence on the condensation rate at the free surfaces.

It is not possible to reproduce experimentally in full scale the whole ECC injection process, starting from the injection nozzle up to the inner downcomer, considering the various two-phase flow regimes. Therefore, reliable numerical simulations are required. Two-phase PTS constitutes one of the most challenging exercises for a Computational Fluid Dynamics (CFD) simulation. Presently available CFD tools are not yet able to reproduce all the separate phenomena taking place in the cold leg and the downcomer during the ECC injection, let alone an accurate simulation of the whole process. Improvements of the two-phase modelling capabilities have to be undertaken to qualify the codes for the simulation of such flows. A really accurate simulation of all the phenomena that occur in the scenario will only be possible in the future after a step-by-step improvement of the quality of the forecasts is achieved. However, a reasonable prediction of the most important phenomena based on CFD may be reached in a short or medium term and the use of CFD in industrial studies related to PTS is already possible with some limitations.

2 Selected activities and results of the European project NURESIM

The aim of the Work Package on Pressurized Thermal Shock in the NURESIM project was to check the current capabilities of CFD models for the simulation of PTS phenomena and to improve these capabilities. The French NEPTUNE_CFD code was the reference CFD code for NURESIM. It was implemented into the NURESIM platform. However, the modelling approaches are similar in most of the CFD codes and the main emphasis of model improvements is connected with the improvement of closure models which are mostly independent of the code used. Benchmarking between different codes
A database on momentum and heat transfer for condensing stratified steam-water flows was generated by Lakehal et al. (2008) [10] based on Direct Numerical Simulations (DNS). From the database correlations were obtained, which can be used in CFD-models for stratified flows. The proposed models were implemented into the NEPTUNE_CFD code and used e.g. for simulations of the DCC. The so-called Large Interface Method (LIM) was developed in the frame of NURESIM by Coste et al. (2007) [11]. In this model three neighboring mesh cells are considered to reduce the effect resulting from the position of the interface in relation to the numerical mesh. The method was implemented into the NETUNE_CFD code and simulations on stratified flow for a horizontal channel (Lavéville and Coste, 2008 [12]) and COSI data (Jamicot and Bestion, 1993 [13]) showed promising results (Coste et al., 2008 [14]).

Different experimental data for stratified flows with and without mass transfer were used to validate the models for momentum, heat and mass transfer used in CFD codes. Bartosiewicz et al. (2007) [15] did simulations on data published by Thorpe et al. (1969) [16] aiming on the generation of waves on a free surface due to the Kelvin-Helmholtz instability and their development. They compared the Euler-Euler Volume of Fluid (VOF) technique implemented in FLUENT with a two-phase model in NEPTUNE_CFD. They showed, that both approaches are applicable. On the other hand surface tension effects are important. For this reason they implemented a surface tension model into NEPTUNE_CFD.

Different approaches for the simulation on slug flow in a horizontal channel were compared by Bartosiewicz et al. (2008) [17]. Three different codes have been used in this benchmark on data obtained at the HAWAC channel (Valée et al., 2008 [18]). CFX was used with a turbulent two-fluid model in which a special turbulence damping function was implemented in the specific dissipation rate of the turbulent kinetic energy. This allowed a good qualitative representation of the slug dynamics, even though quantitative comparison was less relevant because of difficulties in modelling the inlet instabilities. The VOF approach in its laminar and turbulent form was also tried out with the FLUENT code and was found to be inappropriate for those conditions due to the high velocity slip between phases. Moreover, NEPTUNE_CFD was successfully applied with the above mentioned implemented LIM allowing free surface location and the computation of momentum transfer across this interface. With this new model, promising qualitative and quantitative results were obtained, at least for the occurrence of the first slug. Similar results were obtained in a benchmark exercise done by Terzuoli et al. (2007) [19].

Horizontal, stratified flows of sub-cooled water with condensation of saturated dry steam along the water surface were simulated by Scheuerer et al. (2007) [20] and Galassi et al. (2009) [21]. The experimental data for assessing the accuracy of the CFD results have been obtained in the LAOKOON test facility (Goldbrunner et al., 2000 [22]). Co- and counter currents flows with energy and mass transfer at the free surface have been simulated using CFX and NEPTUNE_CFD. In both codes the Euler-Euler two-fluid model concept was applied. For the CFX simulation the influence of using a homogeneous two-phase model versus a full two-fluid model was also examined. The CFX study investigated mainly the influence of two-phase flow model parameters, like interfacial length scales and interfacial heat transfer coefficients. The NEPTUNE_CFD calculations were run with more advanced models for the interfacial drag coefficient and interfacial energy transfer. For instance, the interfacial energy transfer models were based on the surface renewal theory, and different approaches to calculate the interfacial area density were studied. Both CFD codes showed qualitative agreement with the data, but tend to under-predict the temperature in the water layer which is an important quantity to assess the effects of PTS. Unfortunately the LAOKOON data are not very detailed regarding inlet and outlet boundary condition, which makes a more thorough investigation of the discrepancies between predictions and data difficult.

Due to the high temperature differences between the injected water and the two-phase mixture, DCC plays an important role. Simulations were done related to experimental data available for the flooding of a steam-filled pipe from one side (Strubelj and I. Tiselj, 2007, 2008 [23, 24]) with possible condensation induced waterhammer. The main parameters of the simulations compared with the experiments are liquid temperature rises in several points along the pipe. The final liquid temperatures at the measuring points do not depend on the details of the slugging and bubble condensation, but rather on the mixing, which was caused by all the slugs that appeared in the pipe during the transient. The water heat up due to condensation gave some information about the suitability of the applied condensation model. The Large Interface Method by Coste et al. (2007) [11] mentioned above, predicts time evolution of temperatures and final temperatures on measuring points better than models based on the Hughes-Duffy correlation for condensation and the bubble/droplet drag force model.

A small part of the investigations in frame of the NURE-SIM-PTS work package focused on the DCC phenomenon in condensation pools. New experimental data on steam injection into a pool of cold water via a thermally insulated DN200 blowdown pipe (POOLEX-Experiment, Turskanen et al., 2008 [25]) was obtained. Condensation took place only at the steam-water interface near the pipe outlet. Since very low steam flow rates (1.0 ... 1.5 g/s) were used, the steam/water interface remained steady close to the pipe outlet. Simulations with the Hughes-Duffy based DCC model of the NEPTUNE_CFD code indicated two Orders of magnitude higher condensation rates than the experiment. This overestimation was reduced by one order of magnitude by decreasing the numerical truncation parameter and by disabling the residual droplet handling. By implementing the DNS-based model of Lakehal et al. (2008a) [26] the heat transfer coefficient reached the same order of magnitude as indicated by the experiments. More stable transfer rate values were also attained. However, uncertainties prevail in the experimental and simulation results as the presence of non-condensables has not been taken into account.

Simulations on the integral two-phase PTS case were done based on COSI-Experiments (Coste et al., 2008 [14]). The COSI facility represents a 1/100 volume scaled model of the cold leg of a French 900 MW PWR under LOCA thermal hydraulic conditions. Measurements include temperature profiles at various axial positions in the pipe. The calculated water level, the liquid heat up in the cold water injection region and the global condensation were generally within a reasonable range, but there are also some deviations e.g. of the upstream temperature profiles.

3 Conclusions and recommendations for further CFD-model improvements

Simulations of the considered two-phase PTS case include different local flow situations. As shortly discussed in Section 2 some of these local flow situations can be already captured...
quite well by presently available CFD codes. However, still many open questions exist. The most important open questions are connected with the coupling of the different flow regions as well as the coupling of the single effects determining the resulting flow in these regions. This concerns e.g. the treatment of gas or liquid phases which may occur simultaneously as continuous phase and disperse phase in one flow domain. Despite some clear progress achieved during NURESIM further R&D work is required.

For further improvement of the CFD-code capabilities for the two-phase PTS case new well-instrumented experimental data are needed. This concerns single effect experiments as well as integral ones. In single effect experiments the considered phenomenon should be separated as much as possible from other phenomena to obtain detailed data on micro-scale phenomena. A variation of the experimental parameters is important to investigate their influence on the considered phenomenon.

Detailed experimental data are needed especially for the following micro-scale phenomena:

- Condensation at the surface of a sub-cooled liquid jet in steam environment,
- Turbulence production and bubble entrainment below the jet,
- Condensation at the free surface and turbulent mixing in a stratified flow.

Data are required in a high resolution in space and time for the whole domain of interest. Ideally local and time-dependent information on the following parameter has to be measured:

- the structure and the interfacial area density between the phases,
- mean velocities of both phases,
- velocity fluctuations (turbulence parameter),
- temperature and pressure.

Integral experiments, which reflect the PTS flow situations, are important for testing the interaction between all the sub-models. Preferably the same variables as listed above should be measured at many local positions. The data from integral experiments have to be used for the validation of the CFD-codes for the two-phase PTS situation. This is important for the use of the future application of CFD codes for nuclear safety analyses on the PTS issue. Such an integral experiment programme (TOPFLOW-PTS) is conducted outside the NURESIM framework, being financed by EDF, AREVA-NP, CEA, IRSN, PSI, ETH and FZD.

More flexible CFD-models would allow switching between different approaches within one flow domain but at the different local flow situations observed in the two-phase PTS case. Examples for such model approaches are the Large Scale Simulation (LSS, Nícméan et al., 2009 [9]) which should allow the application of a two-fluid model for dispersed flows and Interface Tracking Methods for large surfaces and the Scale Adaptive Simulations (SAS) which allow the simulation of large eddies while modelling the turbulence at the unresolved scales. While LSS modelling should be envisaged for use in the far future, SAS models already exist and should be validated for the flow situations typical for PTS. In addition the applicability of LES should be investigated. The same strategy, referred to as LES by Lakehal (2008) [27] is under validation for various problems, including the COSI test case of Coste and Pouvreau (2007) [11].

Beside turbulence also condensation has a considerable effect on the final thermal loads on the RPV wall. Condensation and heat transfer models for the different flow configurations (surface of the jet, horizontal flows which can be flat or wavy) have to be developed. Also the influence of condensation on the turbulent field should be a topic of future R&D work.

For modelling the transfer of mass, momentum and energy at a free interface it is recommended to test and validate new strategies such as the Large Interface Method to allow mesh convergence with a reasonable coarse grid (Coste et al., 2007). In frame of the NURESIM project it was shown, that aside from information obtained by experimental investigations, also DNS can provide information to build correlations for coarser CFD simulation not resolving the specific phenomenon. Further effort to implement, test and validate the results obtained for interfacial transfers for free surfaces with DNS should be made.

Bubble entrainment due to jet impingement or caused by waves has to be modelled on a physical basis. Up to now it is mainly determined by numerical effects. This future R&D work is connected with the more general modelling of transitions for gas (liquid) being the continuous phase and gas (liquid) being the dispersed phase.

Finally future R&D work has to be directed to the validation of the interaction between the single models based on well instrumented integral experiments. In the near term, one may envisage a simplified treatment of two-phase PTS transients by using a two-fluid model in a RANS approach (k-e model in both liquid and steam), neglecting some effects which are not yet controlled like the bubble entrainment and the possible effects of waves on the free surface. A better modelling of interfacial transfers of heat and mass at the free surface allowing convergence with a reasonable coarse mesh is still required to be able to predict the minimum liquid temperature entering the downcomer with a sufficient accuracy. It is very likely that neglecting entrained bubbles and interfacial waves leads to conservative predictions since both phenomena increase condensation and mixing.

Acknowledgments

The NURESIM project is partly funded by the European Commission in the framework of the Sixth Framework Program (2004–2006). (Received on 17 March 2009)

References


4 Lucas, D. (Ed.): Review of the existing data basis for the validation of models for PTS. 6th Euratom Framework Program NURESIM,

Kerntechnik 74 (2009) 5–6

241


The authors of this contribution

D. Lucas,
Forschungszentrum Dresden-Rossendorf e.V. (FZD)
Institute of Safety Research
P.O. Box 51 01 19, 01314 Dresden, Germany
E-mail: D.Lucas@fzd.de

D. Bestion,
Commissariat à l’Énergie Atomique (CEA)
Centre d’Études Nucléaires de Grenoble
17 Rue des Martyrs, 38054 Grenoble, France

P. Coste, J. Pouvreau, Ch. Morel
Commissariat à l’Énergie Atomique
A. Martin, M. Boucker
Électricité de France

E. Bodele, M. Schmidtke
Forschungszentrum Dresden-Rossendorf e.V.
M. Scheuereer
Gesellschaft für Anlagen- und Reaktorsicherheit
B. Smith, M. T. Dhotre, B. Niceno
Paul Scherrer Institute
M. C. Galassi, D. Mazzini, F. D’Auria
Università di Pisa

Y. Bartosiewicz, J.-M. Seynhaeve
Université Catholique de Louvain La Neuve
I. Tiselj, L. Šrúbelj
Joel Stefan Institute
M. Ilvonen
VTT Industrial Systems
R. Kykri-Rajamäki, V. Tanskanen, M. Paustinen, J. Laine
Lappeenranta University of Technology

You will find the article and additional material by entering the document number KT110041 on our website at www.nuclear-engineering-journal.com