MODELLING, SIMULATION AND EXPERIMENTS ON BOILING PROCESSES IN PRESSURIZED WATER REACTORS

E. Krepper[1], R. Rzehak[1], U. Hampel[1], R. Hampel[2], C. Schneider[2], Th. Frank[3], J. Fröhlich[4], A. Hurtado[4], E. Koch[4], M. K. Koch[5], I. Otic[6], Th. Sattelmayer[7]
[1] Helmholtz-Zentrum Dresden-Rossendorf, Germany, E.Krepper@hzdr.de
[2]University of Applied Sciences Zittau/Görlitz, Germany
[3]ANSYS Germany
[4]Technical University of Dresden
[5]Ruhr Universität Bochum
[6]Karlsruhe Institute of Technology
[7]Technical University of Munich

ABSTRACT

A collaborative project funded by the Federal Ministry of Education and Research (BMBF) in the framework of the R&D program "Energie 2020+" by four Universities, two Research Centres and ANSYS is coordinated by Helmholtz-Zentrum Dresden-Rossendorf (HZDR). The project is directed towards the development and validation of CFD models of boiling processes in PWR in the range from subcooled nucleate boiling up to the critical heat flux. The work is oriented towards basic research and focused on model development, simulation and experiments on different scales. An important objective is the education and maintaining of competence in nuclear technology.

1. INTRODUCTION

Subcooled flow boiling occurs in many industrial applications where large heat transfer coefficients are required. However, the efficient heat transfer mechanism provided by vapour generation is limited at a point where liquid is expelled from the surface over a significant area. This occurs at the critical heat flux where the heat transfer coefficient begins to decrease with increasing temperature leading to an unstable situation. In this event a rapid heater temperature excursion occurs which potentially leads to heater melting and destruction. For a given working fluid, the critical heat flux depends on the flow parameters as well as the geometry of the flow domain. The verification of design improvements and their influence on the critical heat flux requires expensive experiments. Therefore, the supplementation or even the re-placement of experiments by numerical analyses is of high interest in industrial applications.

In the past, many different empirical correlations for critical heat flux were developed and fitted to data obtained from experimental tests. These have been implemented mainly in purpose-specific 1D codes and applied for engineering design calculations. However, these correlations are valid only in the limited region of fluid properties, working conditions and geometry corresponding to the tests to which they were fitted. Using large lookup tables based on a great number of experiments, a significant range of fluid properties and working conditions can be covered. But this method is still limited to only that specific geometry for which they were developed. Independence of the geometry can only be achieved by the application of CFD methods. Existing CFD models, however, are not yet able to describe critical heat flux reliably. A precondition would be the complete understanding and simulation of boiling as a preliminary state towards critical heat flux.

For engineering calculations, currently the most widely used CFD approach to model two-phase flows with significant volume fractions of both phases is the Eulerian two-fluid frame-work of interpenetrating continua (see e.g. Drew & Passman, 1998). In this approach, balance equations for mass, momentum and energy are written for each phase, i.e. gas and liquid, separately and weighted by the so-called volume-fraction which represents the ensemble aver-aged probability of occurrence for each phase at a certain point in time and space. Exchange terms between the phases appear as source terms in the balance equations. These ex-change terms consist of analytical or empirical correlations, expressing the interfacial forces, as well as heat and mass fluxes, as functions of the average flow parameters. Since most of these correlations are highly problem-specific, their range of validity has to be carefully considered and the entire model has to be validated against experiments.

An application of continuing high interest is the thermal hydraulic flow in the core of a nuclear reactor. Here high heat fluxes have to be removed from the fuel rods. In a boiling water reactor al-ready at normal operation conditions boiling occurs. At certain conditions necessarily be excluded violation of the heat flux the flow in the hot channels develops to annular flow. The fuel rod surface is cooled only by a water film which might tear and dryout occur. In a pressurized water reactor at much higher pressure boiling occurs in the form of subcooled boiling described by bubbly flow. With increasing heat flux the number of steam bubbles created at the rod surface increases, the bubbles grow together and departure of nucleate boiling occurs.

The initial states of the second scenario can be described as subcooled flow under bubbly flow conditions. For this type of boiling already two decades ago a wall boiling model based on a heat flux partition algorithm was proposed by Kurul and Podowski (1991). The model was validated and calibrated by several authors. However the further development of this approach stagnated, not least due to the lack of suitable experiments. High pressure, high temperature, narrow channels and small expected sizes of steam bubbles represent significant challenges for measurements.

2. A COLLABORATIVE PROJECT – OVERVIEW

A collaborative project funded by the Federal Ministry of Education and Research (BMBF) in the framework of the R&D program "Energie 2020+" by four Universities, two Research Centres and ANSYS is coordinated by Helmholtz-Zentrum Dresden-Rossendorf (HZDR). The project is directed towards the development and validation of CFD models of boiling processes in PWR in the range from subcooled nucleate boiling up to the critical heat flux. The work is oriented towards basic research and focused on model development, simulation and experiments on different scales.

Small scale tests performed at the University of Applied Sciences Zittau/Görlitz explore the microscopic phenomena of boiling processes. To this end, new measurement techniques like high speed and infrared cameras, micro PIV, holographic interferometry and optical coherence tomography (OCT) are applied. The latter method was developed by the Medicine Faculty of the TU-Dresden and applied up to now only in medicine (see Section 3.1 and 3.2). Applying holographic measurement techniques at the TU-Munich at small scale test facilities operated with a refrigerant, the effect of turbulence and secondary flows on CHF phenomena could be shown (see Section 3.3).

At the other end of the range of investigated scales, a rod bundle test is under construction at the large scale test facility TOPFLOW in Rossendorf (Section 6). The instrumentation of this test with fast X-ray tomography will deliver nonintrusive data with high temporary and spatial resolution suited for CFD model validation. The effect of PWR-specific coolant additives on boiling phenomena is investigated at the experimental facility SECA at TU-Dresden.

In the theoretical field the suitability of RANS turbulence models for CFD describing the two phase flow phenomena in rod bundle geometries are investigated in the Institute of Fluid Mechanics of the TU-Dresden by comparing it to Large Eddy Simulations, which is discussed in Section 5.

In CFD models for boiling, the interfacial area is an essential parameter which can be described conveniently by a population balance approach. The coupling of the wall boiling model with the multiple bubble size group approach (MUSIG) (Krepper et al. 2013) was realized in close cooperation between ANSYS and HZDR: the coupled algorithm was implemented in CFX by ANSYS and validated by HZDR (Section 4). For realistic modelling of wall boiling the heat conduction in the heated wall and the superheating of vapour have to be considered.

CFD simulations were used to improve the results of hot channel codes in the Karlsruhe Institute of Technology (Section 7.1). At the Ruhr University Bochum boiling models implemented in lumped parameter codes are improved for the simulation of RPV cooling from outside in an accident case (Section 7.2).

3. SMALL SCALE EXPERIMENTS

3.1 Work at the University of Applied Sciences Zittau/Görlitz

For investigation of nucleate boiling under forced convection, a test facility was constructed at the University of Applied Sciences in Zittau&Görlitz allowing to apply different optical methods. The working fluid is demineralized water under atmospheric pressure. In the rectangular test section with 28 mm side length nucleate boiling is generated on different geometries. Input conditions cover velocities of 0.1 to 2.0 m/s and subcooling from 2 K to 40 K so that a mass flow of 100 – 2100 kg m^{-2} s^{-1} and a heat flux of 7 – 115 W cm^{-2} can be realized. For the investigation of single bubble evaporation an optically transparent and electrically conductive indium tin oxide (ITO) surface deposited on a borosilicate substrate was used containing an artificial nucleation site. High speed video and infrared techniques were applied.

The ITO layer has a surface resistance of 20 Ohm and provides optical transmission up to 1400 nm. These properties enable the observation of growing bubbles through the heated surface. The ebullition cycles of the bubbles are captured with a frame rate of 13.15 kHz for a sequence of 20,000 frames. To analyze this huge amount of data, an evaluation algorithm was developed for digital image processing (DIP) (Schneider et al. 2012). During this procedure, the images are pre-processed to get binary images for the bubble measurement. The bubble growing and detachment was measured with a temporal resolution of 0.076 ms and a spatial resolution of 25 µm/pixel. For variation of subcooling, flow velocity and heat flux the average departure diameter varies between 0.6 mm and 2.6 mm and the maximum diameter between 1.0 mm and 2.8mm. The smaller departure diameter compared to the maximum diameter results of the increasing condensation rate on the cap because of reaching the subcooled liquid during evaporation on the wake of the bubble. Furthermore, the images are used to apply a particle image velocimetry algorithm for the measurement of the velocity field of growing and condensing bubbles. In the velocity fields of a condensing bubble acceleration is visible between the frames (see Fig. 1). It is recognized that the rapidly changing shape of the bubble induces large local velocities. With the methods applied it is possible to extract necessary parameters for the development of microscopic boiling models for the simulations.

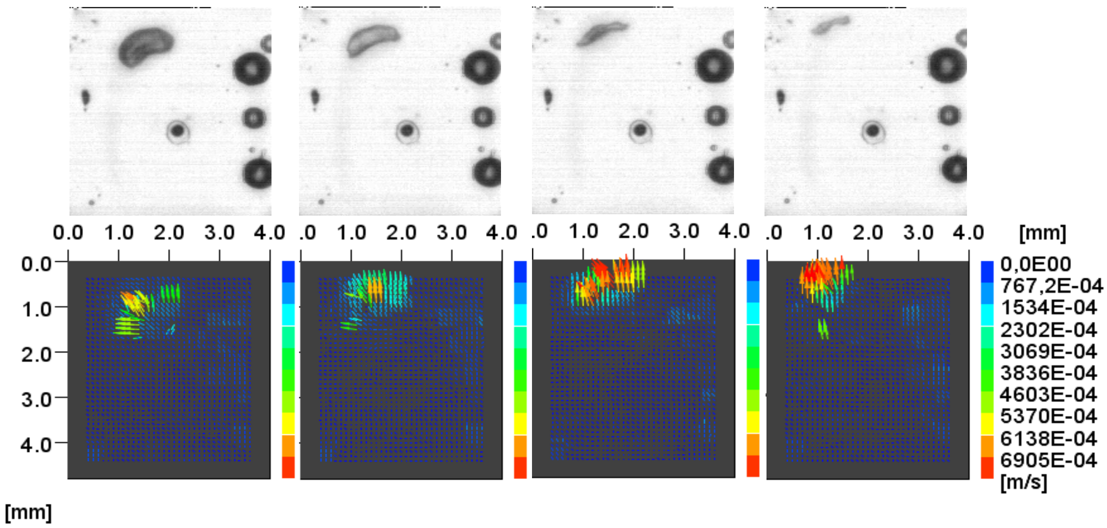


Figure 1: Analysis of bubble condensation in the subcooled vertical up flow. Top: Images of bubble condensation from high-speed camera observed through the backside of the heated ITO-surface (time step 0.228 ms). Bottom: corresponding two dimensional vector fields of local velocity.

3.2 Optical coherence tomography

Optical coherence tomography (OCT) is a relatively new imaging technique, which is already established in biomedical applications. OCT is a contactless and high resolution imaging technique based on white light interferometry utilizing broadband light sources in the near infrared (NIR) range. At the Medicine Faculty of the TU-Dresden the OCT was developed and applied for three-dimensional and two-dimensional time resolved imaging of nucleate boiling on heated surfaces on a microscopic scale with high spatial (<10 μm) and temporal (>25 frames per second) resolution (Meissner et al. 2012). The method was applied at the small scale facility in Zittau and uses the construction of the borosilicate glass plate coated with an optically transparent and electrically conductive indium tin oxide (ITO) layer with a thickness of approximately 100 nm as heating surface. The combination of these two properties allows optical inspection of the nucleate boiling from the backside by OCT focused on the formation, growth and detachment of single bubbles. Figure 2 presents an OCT image sequence of nucleate boiling over a time period of 71 ms. each single image consists of 440 depth-scans corresponding to a lateral image size of 3.4 mm and the acquisition time of each image was 4.2 ms at a frame rate of 238 Hz.

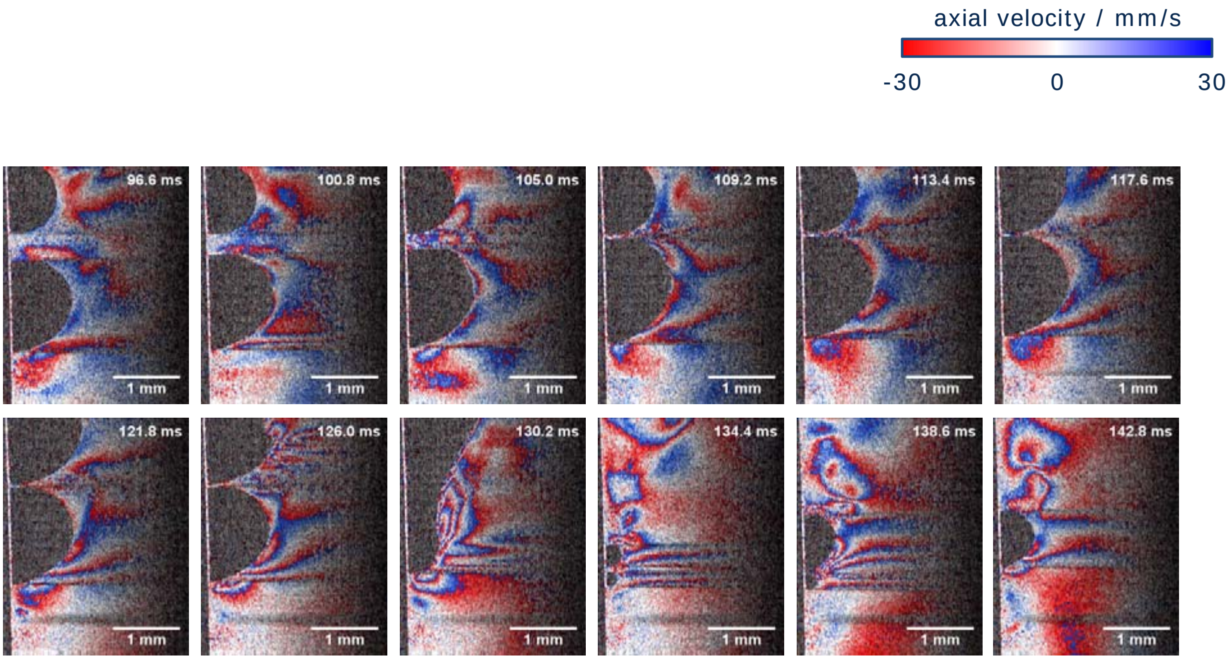


Figure 2: Image series of nucleate boiling recorded with OCT

3.3 Contributions of the Technical University of Munich

At the Institute for Thermodynamics of the Technical University of Munich, a small scale test facility for investigation of boiling processes was constructed. Special interest was given to the effects of turbulence and secondary flows on the boiling processes and the mechanisms during the CHF transient.

The boiling experiments were conducted in a small scale test facility with vertical, rectangular channel with one heated wall in subcooled flow at differing flow rates and fluid subcooling over the entire boiling curve. 3M Novec 649, a low boiling carbohydrate was used as working liquid, making boiling regimes up to fully developed film boiling accessible. Turbulence or secondary flows were added through different flow inserts at the channel inlet.

A wide range of measuring techniques was implemented and further developed to create a broad data base of the heat fluxes, vapour and temperature distributions and the movement of both phases. These techniques included arrays of thermocouples, high-speed photography, Particle Image Velocimetry, fiber optical microprobes as well as Digital Holographic Interferometry and Interferometric Velocimetry. Measurements of turbulence intensity and vorticity in cold flows showed the penetration depths of the inserts. These were shown to be significantly higher for inserts creating longitudinal vortices than for perforated plate type inserts creating near isotropic turbulence (Bloch et al. 2012a).

Boiling experiments showed that the use of flow inserts can strongly enhance the overall heat flux in boiling processes, yielding increases in critical heat flux of up to 90 % in comparison to experiments at identical conditions without inserts (Bloch et al. 2011, 2012a, b). The effects of the inserts could be shown to be mainly based on better mixing and vapor removal in the channel, resulting in more subcooled liquid being transported towards the heater surface. At lower subcoolings, the larger thickness of the vapor layer limits the accessibility of the heater surface to turbulence effects, and the smaller difference in enthalpy between the bulk liquid and the liquid near the heater surface reduces the effect of liquid from the bulk being transported towards the heater. Fig. 3 shows the changes in vapor distribution in the channel from the empty channel case to a twisted tape insert creating longitudinal vortices and an orifice type insert (left to right) at boiling regimes close to CHF (wall superheat 1 K below CHF). Special regard was also given to measurements close to and during the critical heat flux transition, with the aim of improving currently used mechanistic CHF models. The results showed larger slug-like vapor agglomerations appearing near the heater surface at nearly constant frequencies, independent of fluid subcooling and inserts. PIV and HIIV measurements of the velocity fields around such agglomerations showed a significant increase in fluid velocity towards the heater surface in the gaps between the agglomerations. Holographic interferometry revealed the liquid in the gaps to be at roughly the same subcooling as the bulk flow, confirming the importance of these gaps or wetting fronts for the heat transfer in regimes close to CHF as given in the interfacial lift-off model.

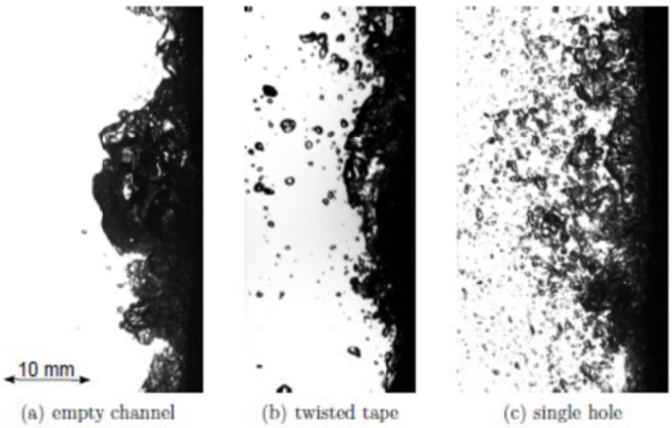
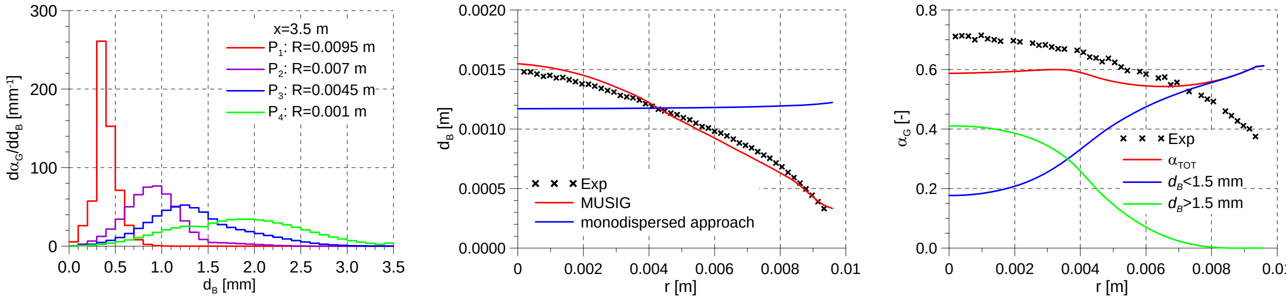


Figure 3: Flow regimes close to CHF for channel without inserts and with a twisted tape and a single-hole insert (Fluid subcooling 9 K, Flow velocity 0.6 m/s)

For the CHF transient, it could be shown that while overall gas production is reduced at CHF, the vapor phase becomes concentrated near the heater surface, with size and speed of vapor structures increasing. This indicated that the actual effect leading to CHF is rather an effect of bubble crowding in the gaps between larger agglomerations than an actual lifting of the interface.

4. MODEL DEVELOPMENT

For CFD modelling of boiling the interfacial area is an essential parameter for the behaviour of the two phase flow, which can be described most detailed by a population balance approach (MUSIG). The coupling of the wall boiling model with the multiple bubble size group approach (MUSIG) was realized in close cooperation with ANSYS and HZDR. The coupled algorithm was implemented in CFX by ANSYS and validated by HZDR. Fig. 4 shows the application to DEBORA boiling tests (Krepper et al. 2013). The inhomogeneous MUSIG approach enables the consideration of the dependency of bubble forces on the bubble size. So the increase of the bubble size by coalescence after leaving the heated surface and the core peak volume fraction profile observed in the experiments could be described by the simulations. The developed models were applied for validation of the OECD –Benchmark of the NUPEC PSBT experiments ((Krepper & Rzehak 2012). Further model development validating models for bubble fragmentation and coalescence are under the way.



a) Bubble size distribution at different distanced from the heated wall

b) Radial profile of the bubble size

c) Radial profile of the gas volume fraction

Figure 4: CFD simulation applying the inhomogeneous MUSIG model approach

For a realistic modelling of wall boiling the heat conduction in the heated wall and the superheating of vapour have to be considered.

5. THEORETICAL MODEL DEVELOPMENT SIMULATING TURBULENT BUBBLY FLOW

At the Institute of Fluid Mechanics of TU Dresden numerical simulations are performed for single phase and multiphase flow in order to advance RANS models, work which is conducted in collaboration with HZDR.

Applying an Immersed Boundary Method for the simulation of a bubble-laden upward turbulent channel flow aspects of two phase flow are investigated. The bubbles are modelled as spherical solid objects of fixed shape which is justified for small bubbles in contaminated water. The bubble-bubble and bubble-wall interaction is accounted for by an appropriate model based on a repulsive potential. First results in (Santarelli & Fröhlich 2012) show the uneven distribution of bubbles for large bubbles accumulating in the channel centre and small bubbles accumulating near the walls known from literature. The large amount of data are presently evaluated by means of various statistical quantities such as spatial correlation functions and the bubble pair correlation function.

A further topic of interest is the turbulence modelling of the flow in a narrow channel like a hot channel of a fuel element. Applying LES to compute the single phase flow in this configuration at high Reynolds number, the flow instabilities were investigated. Figure 5 shows the calculated secondary flow obtained compared to the corresponding calculation performed at HZDR applying the BSL Reynolds Stress Model. Detailed investigations at HZDR demonstrated that these secondary flows have a substantial influence on the cross sectional distribution of the gas volume fraction in the corresponding multiphase flow (Krepper & Rzehak 2012).

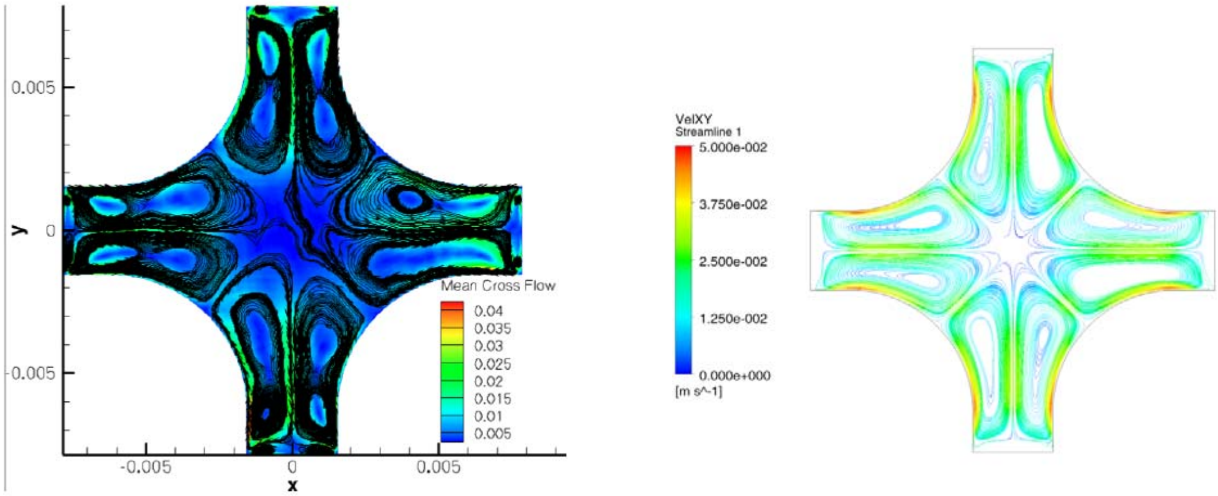


Figure 5: Comparison of the secondary flows in the narrow channel of the NUPEC PSBT Benchmark. Left: Result from LES visualized by means of streamlines a contour plot of the magnitude of the secondary flow. Right: Streamlines of the secondary flow obtained with the BSL Reynolds Stress model coloured with the magnitude of the secondary flow.

6. ROD BUNDLE TESTS AND MODEL VALIDATION

In the HZDR a rod bundle test is constructed. 3 × 3 rods with a rod diameter of 9.6 mm and a rod distance of 12.6 mm are arranged in a tube with an inner diameter of 50 mm (s. Fig. 6).

Different tests applying different measuring techniques were performed. To assess the flow distribution in the bundle, a series of PIV measurements for single phase flow were conducted. At a first two phase investigation, gamma densitometry was applied. Finally the method of the ultrafast X-ray tomography was developed and applied for the rod boiling test.

6.1 Single phase tests

6.1.1 The experiments

These experiments were carried out in cooperation with the Texas A&M University (Dominguez-Ontiveros et al. 2012). For PIV, a free and undistorted optical access to the flow channel is substantial. Therefore, the titanium rods were replaced by acrylic rods with same diameter and the acrylic tube was covered with an acrylic correction box (see Figure 7). To achieve optimal optical conditions a refractive index matching liquid (p-cymene) was used (refractive index: acrylic glass: 1.492, p-cymene: 1.4907). The flow channel as well as the gap between acrylic tube and optical correction box was filled with p-cymene as well. Experiments had been carried out at normal temperature and pressure. Experiments and simulations were conducted for three different inlet volume flow rates: 1.24 l/s, 1.72 l/s and 2.14 l/s.

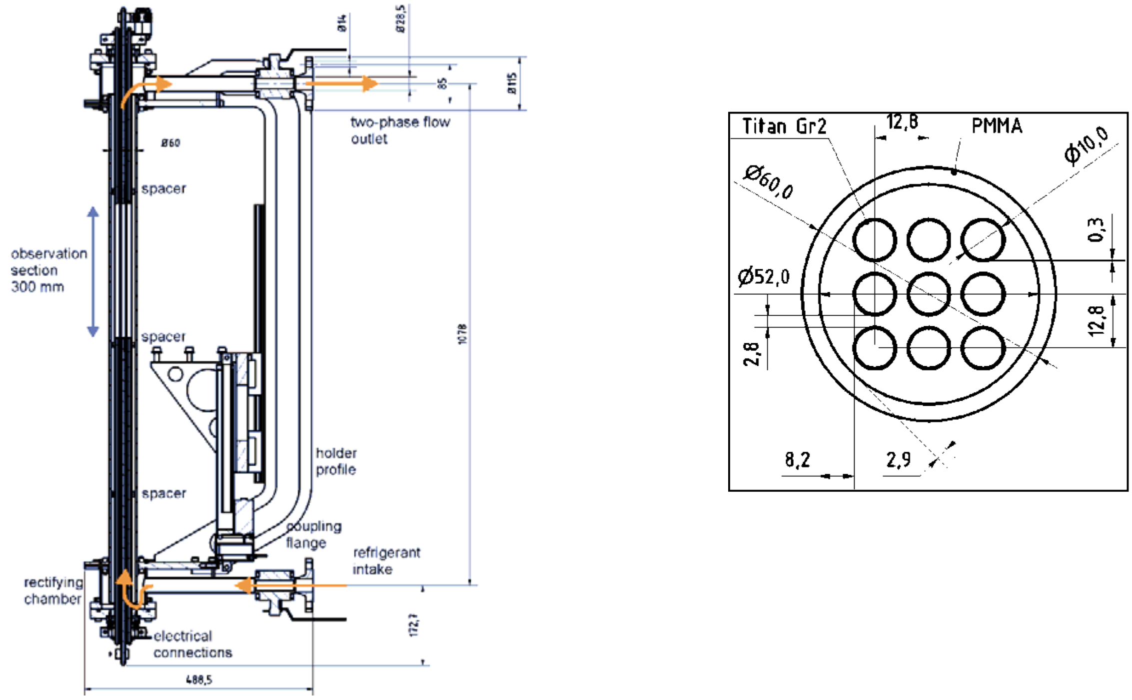


Figure 6: Bundle test and cross sectional view

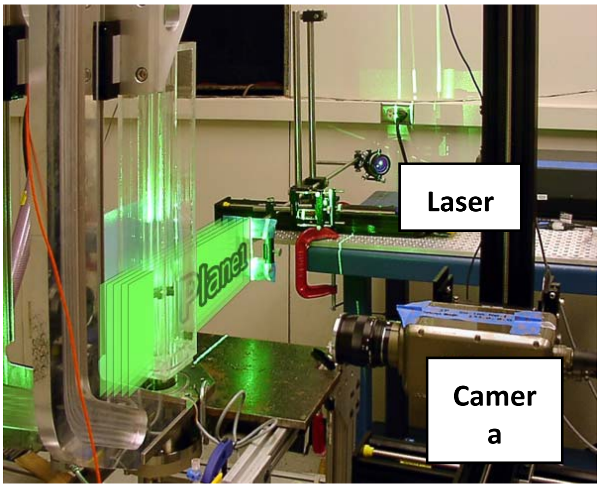


Figure 7: Arrangement of the single phase rod bundle test

6.1.2 CFD analyses of the single phase tests

Turbulent single-phase simulations in a geometrical domain reproducing the test facility (3x3 rod bundle) were conducted applying the CFX code ANSYS-CFX (Lifante et al. 2012). The investigations were aimed to perform an initial study for future analysis of boiling processes in the same test facility. The current CFD simulations allow an assessment of the turbulent flow conditions in this test facility.

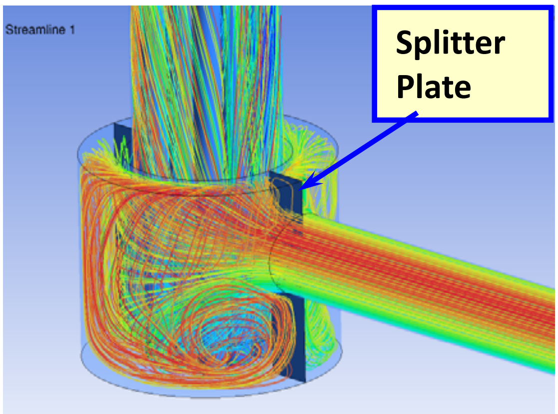


Figure 8: Equipment of the inlet chamber with a splitter plate

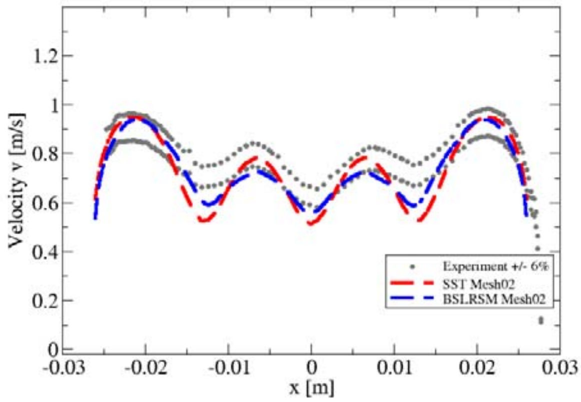


Figure 9: Influence of turbulence modelling on the axial velocity component

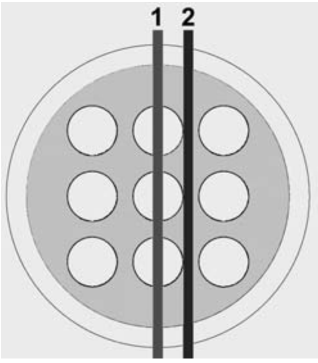
The whole test facility including inlet and outlet region was simulated. Grids with from $6x10^5$ to $8x10^6$ elements were investigated. One of the first results of the preliminary calculations was the emphasis on the experiment to introduce a splitter plane into the inlet chamber to stabilize the inlet flow (see Fig. 8). Otherwise strong oscillations occur which would impede the establishment of defined repeatable boundary conditions. An essential second result was the need of simulating the effect of the grid spacers according to the rod test bundle. The effect of different turbulence models was investigated (see Fig. 9).

6.2 Gas fraction measurement in boiling flows with gamma ray densitometry

For theses boiling tests the refrigerant RC318 was used. Applying gamma ray densitometry the integral time averaged gas fractions along two chord positions through the bundle were determined (Barthel et al. 2012). One of them goes through the central rod row and the other goes through one of the gaps, touching the central rod row (see Fig. 10).

For the experiments a gamma ray densitometer with a 4 mCi 137Cs radioisotope source was used. The gamma ray energy is 662 keV. The detector, which is positioned opposite to the source, is a highly sensitive photomultiplier detector. Both source and detector were equipped with a narrow-beam collimator made of lead, which has a channel width of 2 mm and a channel height of 5 mm. This collimator provides sufficient protection from non-linearity errors originating from scattered radiation. With this setup a 2 × 5 mm² pencil beam of radiation is generated.

The results for the measurement position 2 through the gaps are shown in Fig. 10b. The measurement position was 206 mm above lower end of heating area. The superficial velocity of RC318 was chosen to 0.35 and 0.56 m/s. The sub cooling was varied from 6 to 9 K and the heat flux was adjusted from 85 to 170 kW/m2 in 11 steps. The measured gas fraction strongly depends on the sub cooling at the entrance of the test section and, for measurement position 2, on the heat flux. For position 1 can be assumed, that the data curves are forming a saturation plateau, which has been proven in some extra measurements at lower heating power. The local mean gas fraction at the measured parameters is at a level, which leads to the assumption of a gap to sub-channel gas drift. The dependence of gas fraction on the superficial velocity is less than expected for both measurement positions. Subsequent experiments will deal with the mentioned saturation effects and the assumed gas drift, as well as with gas fraction measurements at lower heat flux.



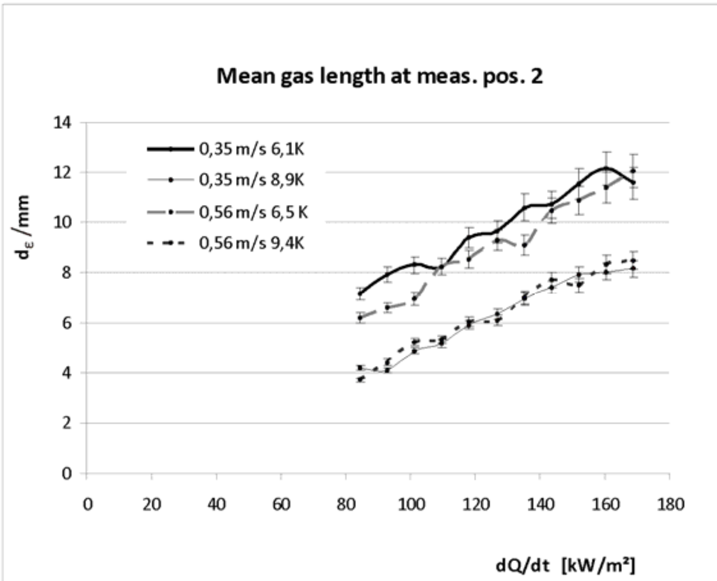


Figure 10: Position of the gamma ray and measured results

6.3 Application of ultrafast X-ray tomography

The ultrafast X-ray tomography is a non-invasive imaging technique, which was developed at HZDR and gives information about the phase distribution in the cross section of the flow with very high temporal and spatial resolution. The working principle of the ROFEX CT-system (ROssendorf Fast Electron beam X-ray tomograph) is shown in Fig. 11. An electron beam of sufficient energy is swept rapidly across a circular X-ray producing target which surrounds the test section, by means of an electromagnetic focusing and deflection system. The X-radiation which is produced in the focal spot passes the object, is attenuated due to density distribution in the object exponentially. Attenuated radiation signals are recorded by a fast detector system which is designed as a circular ring and mounted stationary with some axial offset relative to the focal spot path. From datasets of projections of the object per one revolution of the beam a cross sectional image can be reconstructed using conventional filtered back projection algorithm. Further details of the ROFEX CT system can be found in Fischer et al. 2008.

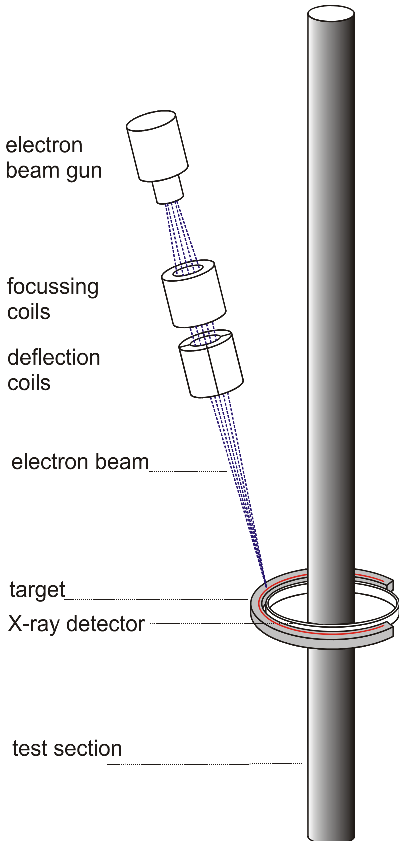


Figure 11: Working principle of the ultrafast CT scanner ROFEX (left) and photography of the special designed system comprising the bundle test section

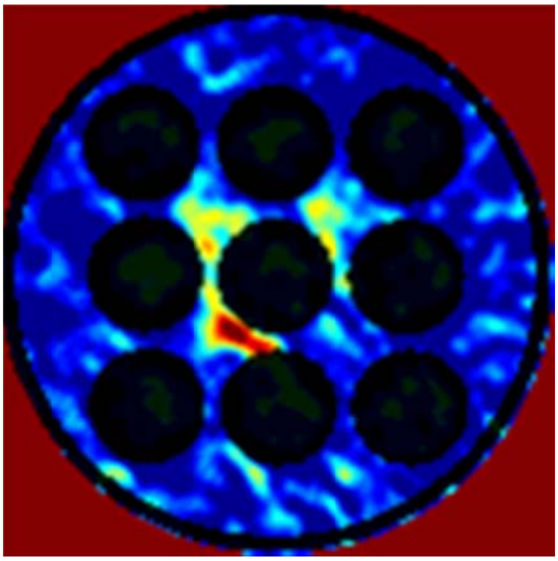


Figure 12: Void fraction distribution determined by ultrafast X-ray tomography

For the boiling experiments in the rod bundle a dedicated CT-scanner based on the ROFEX system was designed, built up and fitted to the bundle test section (Fig. 6, Barthel et al. 2012). The CT system provides sequences of cross-sectional grey value images of the phase distribution in the bundle with a speed of up to 8000 frames per second at a spatial resolution of roughly 0.8 mm. Size of boiling bubbles on rod surface are considerably smaller and phase boundaries cannot be resolved for the small bubbles. Nevertheless, we expect to derive the phase fraction from the grey values and make gas films visible. In a first experiment a parameter setup equal to the gamma ray measurements was chosen. For was acquired with a frame rate of 2000 images per second. Fig. 12 shows the results for an exemplary 0.4 s long scan at moderate heat flux and liquid flow rate.

7. APPLICATION TO MACROSCOPIC PHENOMENA

7.1 Modelling of the void drift phenomenon in sub-channel codes

Void drift is a directional interchannel mixing phenomenon under two-phase flow condition, of which the physical mechanism not completely clarified. Direct measurement on void drift is restricted due to the complex nature of turbulent two-phase flow in narrow channels and more importantly due to the coexistence of void drift with turbulent mixing and diversion cross flow. In the sub-channel code MATRA, mixing effect due to cross flow is directly solved in the lateral momentum equation. Modified equal-volume void drift model based on the assumption of a two-phase system approaching equilibrium void distribution state is applied accounting for the mixing effect due to turbulence mixing and VD. To better understand the two-phase bubbly flow behavior in rod-bundle geometry, in the KIT CFD simulation with the Eulerian two-fluid-model were carried out and a new approach for the consideration of the void drift phenomenon in sub-channel codes was proposed.

7.2 External Reactor Vessel Cooling for a LP-Code

In the course of a severe accident scenario in a nuclear power plant with a core melt and relocation the integrity of the reactor vessel may be preserved by External Reactor Vessel Cooling (ERVC). By flooding the cavity surrounding the reactor vessel the decay heat is removed from molten core material through the vessel wall to water at the submerged vessel surface. In order to simulate the boiling process at the external vessel wall during ERVC in the framework of lumped parameter codes a model based on the CFD boiling model approach was developed. Hence the downward facing orientation of the outer surface of the RPV lower plenum has an influence on the boiling process. Factors resulting from experiments on ERVC realized by Yang 2005 have been added to the heat flux partitioning approach of the CFD boiling model.

8. SUMMARY

In the framework of the three year project seven doctoral thesis and numerous diploma theses were written. Practical trainings were performed at various occasions. All partners met at least once per year and informal meetings of smaller groups were organized when needed. Furthermore, workshops for external participants were organized.

As a result of the project several new experimental facilities are now available and new measurement techniques were developed. Their usefulness has been demonstrated in several papers which just describe the start of their application. Furthermore, new ideas for model developments in the area of multiphase flows were realized. The results of the project will now be applied to further systematic investigations of the detailed boiling processes.

9. ACKNOWLEDGEMENTS

The work was funded by the German Federal Ministry of Education and Research under the contract number 02NUK010.

10. REFERENCES

Barthel, F., R.Franz, E. Krepper and U. Hampel, 2012. Experimental studies on sub-cooled boiling in a 3 x 3 rod bundle, *CFD4NRS-4, OECD/NEA and IAEA Workshop*, 10.-12.09.2012, Daejeon, Korea

Dominguez-Ontiveros, E.E., Y.Hassan, R. Franz, R. Barthel, U. Hampel, 2012. Experimental study of a simplified 3 x 3 rod bundle using DPTV, *CFD4NRS-4, OECD/NEA and IAEA Workshop*, 10.-12.09.2012, Daejeon, Korea

Drew, D. A., Passman, S. L., 1998. Theory of Multicomponent Fluids, Springer.

Fischer, F. et al. 2008. An ultrafast electron beam x-ray tomography scanner, *Measurement Science and Technology*, 19 094002.

Kurul, N. & Podowski, M. , 1991: On the modeling of multidimensional effects in boiling channels, *ANS Proceedings of 27th National Heat Transfer Conference*, Minneapolis, MN

Krepper, E.; Rzehak, R.; Lifante, C.; Frank, T., 2013. CFD for subcooled flow boiling: Coupling wall boiling and population balance models, *Nuclear Engineering and Design* 255, 330-346

Krepper, E.; Rzehak, R., 2012. CFD analysis of a void distribution benchmark of the NUPEC PSBT tests: model calibration and influence of turbulence modelling, *Science and Technology of Nuclear Installations*, 939561

Krepper, E., Rzehak, R., 2011. CFD for subcooled flow boiling: Simulation of DEBORA experiments, *Nuclear Engineering and Design*, 241, 3851– 3866

Lifante, C., B. Krull, Th. Frank, R. Franz and U. Hampel, 2012. 3x3 rod bundle investigations, CFD single-phase numerical simulations, *CFD4NRS-4, OECD/NEA and IAEA Workshop*, 10.-12.09.2012, Daejeon, Korea

Meissner, S., Herold, J., Kirsten, L., Schneider, C., Koch, E., 2012. 3D optical coherence tomography as new tool for microscopic investigation of nucleate boiling on heated surfaces. *International Journal of Heat and Mass Transfer*, vol. 55, p. 5565

Santarelli, C.; Fröhlich J.2012. Simulation of bubbly flow in vertical turbulent channel, *Proceedings in Applied Mathematics and Mechanics*, 26 July 2012

Schneider, C., R. Hampel, A. Traichel, A . Hurtado, S. Meissner, E. Koch, 2011. Experimental investigation of nucleate boiling on heated surfaces under subcooled conditions, *NURETH-14*, Toronto, Ontario, Canada, September 25-30, 2011

Schneider, C., R. Hampel, A. Hurtado, S. Meissner, E. Koch, A. Traichel, 2012. Experimental investigation of bubble behavior under pwr-specific subcooling and flow parameters by optical measurement methods, *Topsafe 2012*, Helsinki, Finland, April 22- 26, 2012

Yang, J., 2005. Development of Heat Transfer Enhancement Techniques for External Cooling of an Advanced Reactor Vessel, Pennsylvania State University, Thesis, December 2005