IAHR Congress, Special Seminar on
"Advances in Industrial Hydraulics and Applications to Energy Production"
Beijing, China, September 17-21, 2001

Analysis of an LWR Sump Cooling Concept

G. Grötzbach, L.N. Carteciano, B. Dorr, X. Jin
Forschungszentrum Karlsruhe GmbH
Institut für Reaktorsicherheit
D-76021 Karlsruhe, Germany
groetzbach@irs.fzk.de

Abstract

In safety investigations of the European Pressurized water Reactor EPR situations are considered in which the core is molten and collects on the core catcher in the sump of the containment. An optional cooling concept was investigated at the Research Center Karlsruhe to remove the decay heat utilizing single phase natural convection in the water above the core melt. The natural convection was investigated by the SUCOS-2D and -3D scaled experiments. Here, results are reported of numerical investigations and interpretations of one of the SUCOS-3D experiments using the computer code FLUTAN. An unexpected temperature distribution is observed in this 3d experiment. Based on the experimental data it can be presumed that one of the horizontal coolers was slightly tilted against the main flow direction. Approximate numerical investigations show that a tilting of only one percent would explain the experimental flow field. In contrast, tilting the horizontal cooler slightly in the circulation direction improves the cooling concept; this could be accurately shown by the new code version which was recently extended from using a finite volume scheme in Cartesian and cylindrical coordinate systems to a new method based on body fitted grids and a coordinate transformation.

1. Introduction

The core catcher in the reactor sump is the final safety barrier after a core melt-down accident. The spreading of the melt and its cooling is investigated in detail world wide, see e.g. in Alsmeyer (2000). In common concepts one is aiming at distributing the heat generating core melt on a large horizontal surface, called core catcher, and cooling it from below. In the development of the European Pressurized water Reactor EPR one aims at achieving a new, improved quality of safety (Bonhomme & Krugmann 1993, Kuczera et al. 1994). Thus, also new concepts for long-term cooling of the core melt in the reactor sump are considered, like freezing, fragmenting and cooling it by leading water through the core melt (Alsmeyer & Tromm 1999) or by covering it with water and cooling it from above (Weisshäupl & Bittermann 1995). The latter optional cooling concept utilizes passive safety features to remove the decay heat from the sump. After the accident, a dry distribution and stabilization of the core melt in the sump region of the reactor is foreseen (Fig. 1). Then cooling of the core melt begins with the water from the in-containment refueling water storage tank. Water cooled heat exchangers and condensers in the reactor sump region remove the heat from inside the containment. The heat is transferred from the core melt to the cooling devices during the first days after the accident by evaporation, and later on by single-phase natural convection and conduction.

The single phase natural convection was experimentally and numerically investigated in the Research Center Karlsruhe. The aim of the experimental SUCOS (SUmp COoling Small) program (Knebel & Müller 1995) was to obtain quantitative results which can be transferred to the prototypic condition in order to make a statement on the feasibility of the single phase sump cooling concept. Two scaled facilities (1:20) were applied in the program: SUCOS-2D (Fig. 2), which represents a two dimensional plane slab (580*275*235 mm) of a simplified reactor sump geometry, and SUCOS-3D (Fig. 3), which is a three dimensional scaled geometry (580*275*1298 mm) of the sump. Water is heated by a copper plate at the bottom of the pool simulating the core melt and cooled by horizontal and vertical heat exchangers in areas where they are protected against vapor explosion consequences.

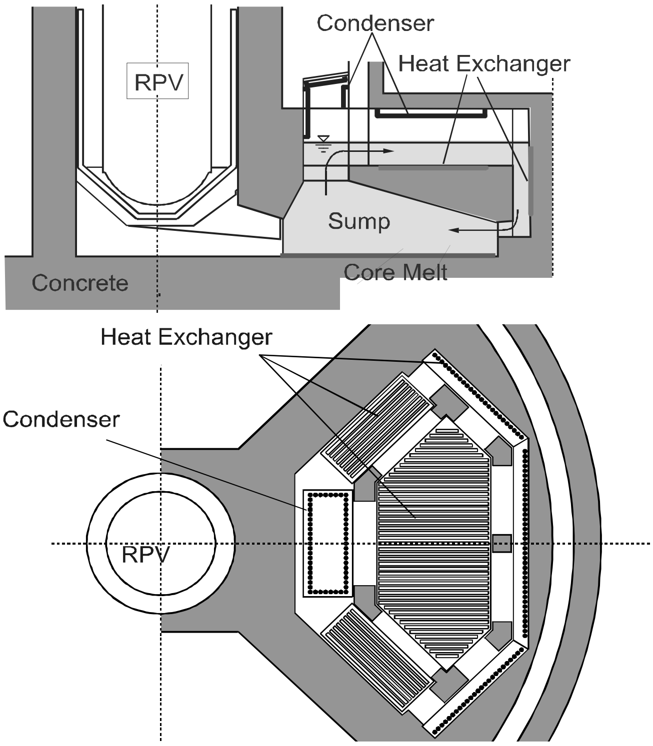


Figure 1. Schematics of the investigated sump cooling concept.

This sump cooling concept was numerically investigated by using the FLUTAN code (Willerding & Baumann 1996). This thermal- and fluid-dynamical computer code is developed in the Research Center Karlsruhe for the numerical analysis of the passive decay heat removal in new reactor systems. It was already extensively validated and applied to analyses of model experiments for the decay heat removal in the fast breeder reactor SNR-300 (Weinberg et al. 1996). It is used here to investigate and interpret numerically the single-phase natural convection in the experiments SUCOS-2D and 3D. The aim of this numerical investigation is to consider the feasibility of the sump cooling concept and to analyze in more detail the experiments.

The first step of the numerical investigation consisted of the simulation and interpretation of a SUCOS-2D experiment (Carteciano et al. 1999). The most important result was that SUCOS-2D cannot be well reproduced by two-dimensional calculations whereas three-dimensional calculations reproduce the experiment quite well. The simulations showed that three-dimensional experiment specific phenomena are significant in the experiments, but these 3d phenomena will be of much less relevance in the reactor sump. Furthermore, the analysis of calculations of a SUCOS-2D experiment gave information on the requirements for the numerical modeling of the geometry and boundary conditions for one of the SUCOS-3D experiments which was the second and final step of the numerical investigation of the single-phase sump cooling concept. This step of analyzing in detail the SUCOS-3D experiment is documented in (Carteciano et al. 2000).

The problems of deducing data for the reactor sump from the results of the model experiments are discussed in (Grötzbach et al. 2000). Transferring the data by scaling laws or CFD-tools is not free of serious problems because the model experiments showed laminar flows, whereas the reactor sump will have turbulent flow conditions (Knebel & Müller 1997). So, reliable turbulence models are required, but purely buoyant flows are still a challenge for any turbulence model, see e.g. Hanjalic (1994, 1999). Also the TMBF, which is explicitly developed for buoyant flows, has up to now only been validated for two-dimensional forced, mixed, and natural convection (Carteciano et al. 1997, 1999b). Thus, the only way which is nowadays often considered to give a better solution for 3d time-dependent flows is to apply Large Eddy Simulation methods (LES). Indeed, there exist already several applications of LES to reactor typical flows (Grötzbach & Wörner 1999).

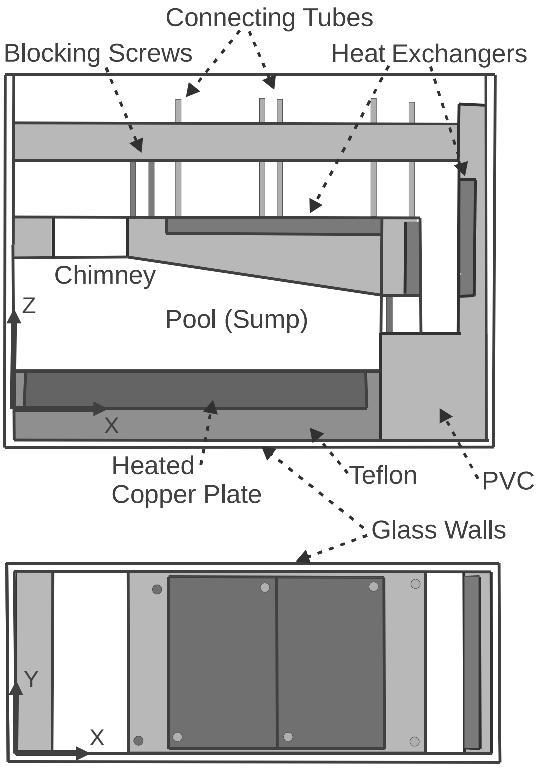


Fig 2. Schematics of the SUCOS-2D test geometry and coordinate system

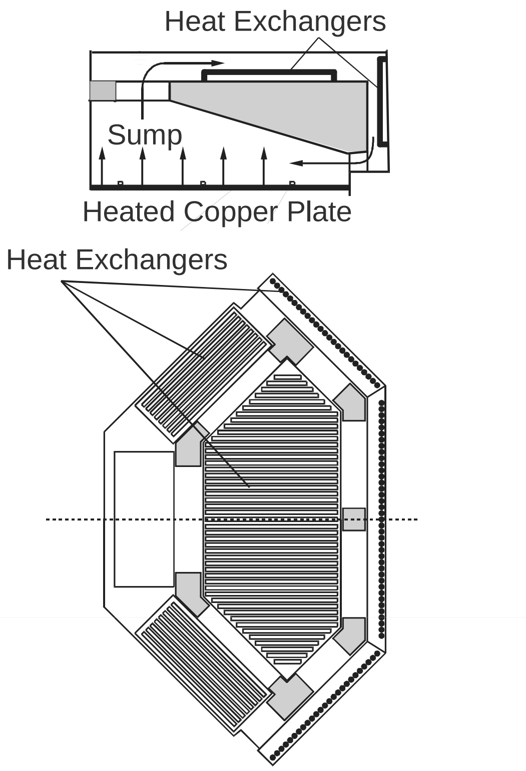


Fig 3 Schematics of the SUCOS-3D test geometry

The objective of this paper is to discuss the most important results of the numerical interpretation of the SUCOS-3D experiment and to conclude on a simple concept improvement. A small geometric distortion which had to be postulated to exist in the experiment and which reduced the efficiency of the cooling concept launched an idea to deduce a minor geometric modification which should increase the convective heat removal. Recent simulation results from the new code version applying non-orthogonal grids (Jin 2001) help to analyze the effect of tilting the horizontal cooler by a small amount in the mean flow direction which should improve and stabilize the cooling conditions for the core melt.

2. FLUTAN Code

FLUTAN is a highly vectorized computer code for 3D fluiddynamic and thermal-hydraulic analyses in Cartesian or cylinder coordinates. It was developed in order to simulate single phase flows with small compressibility. The conservation equations for mass, momentum, energy, and turbulence quantities are discretized on a structured grid by using a finite volume method. A staggered grid is used for the velocities. The discretization of the diffusive terms is performed by a central difference method (CDS). A first order upwind or one of two second order upwind methods (QUICK (Leonard

1979) and LECUSSO (Günther 1992)) can be chosen for the convective terms. A first order implicit Euler-method is used for time discretization.

Several turbulence models are available in FLUTAN. The most important one for buoyant flows is the Turbulence Model for Buoyant Flows (TMBF) which consists of a first order k-ε model in a low-Reynolds number formulation and a second order five-equations turbulent heat flux model (Carteciano et al. 1997). In several benchmarks it turned out that the TMBF in its current development status is a powerful tool at least for forced and mixed convection (Baumann et al. 1997). Special thermal boundary conditions are available in order to simulate different thermal situations like a heat exchanger model and a wall model. A 3d heat conduction model for the structures was developed for the investigation of the SUCOS experiments. This is necessary for simulating solid structures with internal non-uniform transport of heat; it was required to achieve realistic boundary conditions for the fluid domain at the heated copper plate in the SUCOS experiments. The structure temperatures are discretized on an own grid on which the heat conduction equation is solved in all dimensions independent of the solution of the corresponding equation in the fluid domain.

Recently, a new numerical calculation method was developed (Jin 2001) for more accurate computational fluid dynamics in a complex geometry. The method is basing on body fitted grids. It solves the conservation equations in a general non-orthogonal coordinate system which matches curvilinear channel walls. In practical applications the non-orthogonal patched grid is generated by a commercial grid generator. The resulting geometrical data are transformed by means of an interface program to the FLUTAN input format.

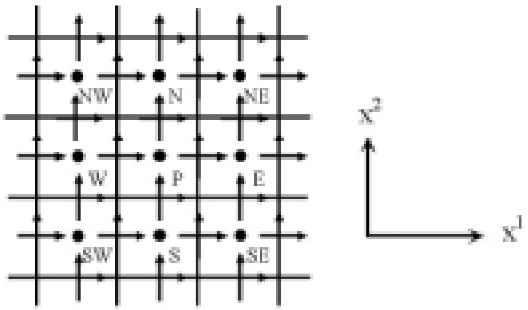


Figure 4: Arrangement of velocities on the transformed staggered grid used with the new finite volume scheme, simplified to 2d.

The conservation equations are transformed from the Cartesian to a general curvilinear system keeping the physical Cartesian velocity components as dependent variables. Using a staggered arrangement of variables, the three Cartesian velocity components are defined on every cell surface, Fig. 4. Thus, the coupling between the pressure and the velocities is ensured even on strongly distorted grids, and numerical oscillations are avoided. The contravariant velocity for calculating the mass flux on a cell surface results from the dependent Cartesian velocity components. The discretization and linear interpolation results in a three dimensional 19-point pressure equation. Treating the cross-derivative terms explicitly reduces the system to the usual 7-point equation. The data structure and solution procedure of this new method is compatible to the one of most codes applying finite volume schemes in Cartesian staggered grids; thus it is also fully compatible to the one used in the Cartesian FLUTAN code version. Therefore, the implementation of the method in FLUTAN was a straightforward activity.

In order to verify the new method, several laminar flows were simulated using orthogonal grids at various tilted space directions and in non-orthogonal grids with varying angles between the coordinate axes (Jin 2001). Among the simulated flow types were several duct flows, transient heat conduction, natural convection in a chimney, and natural convection in cavities. Their results achieve very good agreement with analytical solutions or empirical data. Convergence is also obtained even for highly non-orthogonal grids.

3. SUCOS-3D Investigation

3.1 The test facility

The SUCOS-3D test facility consists of a tank of 580*275*1298 mm (Knebel 1999). The outer walls are 20 mm thick and made of Plexiglas (Fig. 3). The ratio between lengths in the test facility and in EPR is 1:20. A 30 mm thick copper plate heated from below by electric conductors simulates the core melt; it is isolated from the ground, from the walls, and from outside with Teflon. The heating power is scaled according to the volume of the sump as $(1/20)^3$. Plexiglas structures replace the structures of concrete in the prototype. The heat exchangers are slab heat exchangers made of copper; their cooling tubes have a meander form and use water as a cooling fluid. The horizontal heat exchanger is divided in four separate sections: two small and two large ones. The vertical heat exchangers consist of 8 sections: 4 inner and 4 outer sections respectively. Additional coolers in an upright position are present in the test facility. The space above the heated copper plate is called pool, see Fig. 2. The space above the horizontal heat exchangers is called horizontal side area. The space between the vertical heat exchangers is called vertical side area. The vertical channel without heat exchangers connecting the pool and the horizontal side area is called chimney.

Several experiments were performed in SUCOS-3D varying the value of the power input to the copper plate, the arrangement of the heat exchangers, the inlet temperature of the secondary fluid in the heat exchangers T_{cool}, and the level of water. Only temperatures were measured using thermocouples in two planes near the mid section. The experiments are characterized by a so called pool temperature. This is the mean temperature in the pool area which is measured by six thermocouples below the tilted roof of the pool.

In order to achieve optimum use of the numerical simulation and interpretation with FLUTAN, a SUCOS-3D experiment had to be chosen which is consistent with the one already simulated for SUCOS-2D. Therefore, one was chosen in which the horizontal and the outer vertical coolers were in operation, while the internal vertical coolers were not active. The electric heat supply of the heated copper plate amounted to 1,240 W. The inlet temperature of the coolant on the secondary side of the heat exchangers was set to $T_{cool} = 20\ ^0C$ and the flow rate was 20 or 40 g/s. The measured pool temperature was $32.6\ ^0C$.

3.2 Specification of the Computational Model

Only half of the complex but symmetric geometry of the test facility is simulated to reduce the computational effort. According to the experience gained from the numerical analysis of SUCOS-2D, all structures have to be modeled in detail and a very fine grid is necessary for a good resolution of the thin boundary layers near walls and coolers. The mesh size of the grid changes from 8 mm to 1 mm. Ratios of the mesh sizes between two neighboring cells are less than or equal to 2. The 3d grid consists of 691,000 fluid cells and 68,000 structure cells. A first order upwind scheme is used to compute the convective fluxes of enthalpy and momentum. No turbulence model is used because the flow was laminar in the experiment.

The connecting tubes of the coolers, which are present in the horizontal side area are spatially recorded and modeled even if it is expected that they would have a minor influence on the natural convection than in the calculation of SUCOS-2D. The heat losses to the outside through the lateral walls are neglected. The active heat exchangers can be modeled by a heat exchanger model or by pre-setting a distribution of the surface temperature or of the surface heat flux. The calculation of SUCOS-2D showed that it is not necessary to simulate the coolers with the complex heat exchanger model. It is sufficient to give a distribution for the temperature on the surface between fluid and cooler. The linear distribution of the cooler temperature is approximated by a step function prescribing three values for the vertical right coolers. For the horizontal coolers a constant value of temperature is sufficient because the difference of temperatures between the inlet and outlet coolant water is less than 1 K. The prescribed values are determined by means of the experimental data.

In former simulations for SUCOS-2D it was found that the heated copper plate needs special attention (Kuhn 1996). Even the freely developing circulation sense in the fluid domain is sensitive to the thermal boundary conditions used at the upper surface of the copper plate (Grötzbach et al. 1997). There, the problem of using an artificial Neumann or Dirichlet boundary condition was analyzed by calculating the heat conduction in the copper plate. 2d tests showed the surprising result that the copper plate does not ensure a constant heat flux to the fluid, but that it redistributes the heat horizontally in such a strong manner, that the heat flux into the fluid varies along the plate surface by more than +/- 50% of its mean value, Fig. 5. Thus, the thermal conduction in the heater plate is also calculated here. A 3d grid is used for the heated plate; the horizontal grid width distribution corresponds to the one of the fluid region; 5 cells are used in the vertical direction with mesh sizes of 6 mm. The electrical heaters below the copper plate are simulated as a heat flux boundary condition with constant horizontal distribution.

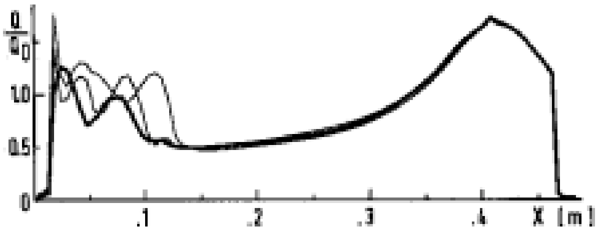


Figure 5: Horizontal distributions of the calculated heat flux Q divided by its mean value Q_0 on the upper surface of the copper plate at three different times in SUCOS-2D.

The simulation was performed on a CRAY J916 with a memory need of 2.7 Gbytes. The transient calculation was preceded by a steady state calculation to obtain an initial flow and temperature field for the transient calculation. The steady state calculation is stopped when an equilibrium in the changes of temperature and in the balance of the heat fluxes is nearly achieved. This happened after 4 h corresponding to 240 h of CPU-time. The transient calculation is performed for a problem time of 227 s with a time step width of 1.0 s. This corresponds to 407 h CPU-time. The system of the pressure equations is solved by the iterative CRESOR method (Borgwaldt 1990), whereas the system of the enthalpy equations is solved by the iterative SOR method.

3.3 Results

A feature, which is inherent to most purely buoyant flows, is its local time-dependence. From large heated plates, the heat is mainly removed by intermittent plume like phenomena. Here, the surface temperature on the copper plate, Fig. 6, is very straggly as it is typically found in Rayleigh-Bénard convection in which the hot plumes rise mainly from the knots of those structures (see the movies in Wörner & Grötzbach 1997). In addition, cold plumes are observed which plunge down through the chimney with a low frequency; this phenomenon was not completely filtered out in the experiment by time-averaging of all data over two minutes. This causes the wall heat flux on the copper plate to change in time, Fig. 5.

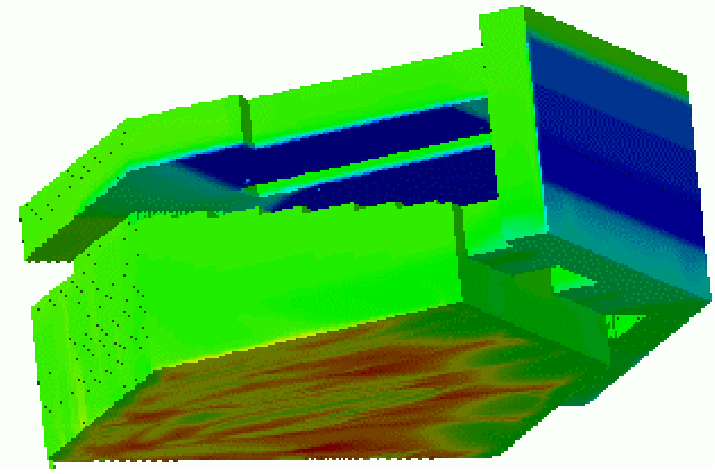


Figure 6: Calculated temperature field on the surface of the structures of SUCOS-3D. View from below on the fluid domain. The dark areas represent the cooler surface temperatures, the straggly surface represents the instantaneous temperature distribution on the heated copper plate.

All following FLUTAN results are from the transient calculation and are time averaged over two minutes, like in the experiment (Knebel 1999). The calculated temperature field is shown in Fig. 7. Despite a careful and detailed 3d modeling of the geometrical and thermal characteristics of the SUCOS-3D experiment, the calculated pool temperature $T_{p,cal}$=29.2 ^{0}C is lower than the measured one $T_{p,exp}$=32.6 ^{0}C. The temperature difference appears to be small but the corresponding relative deviation related to the coolant inlet temperature is with $(T_{p,cal}- T_{p,exp}) / (T_{p,exp}- T_{cool})$ = 34% quite considerable. This difference may decide between having single or two-phase flow conditions after scaling the data up from the model experiment to the reactor sump (Knebel & Müller 1997).

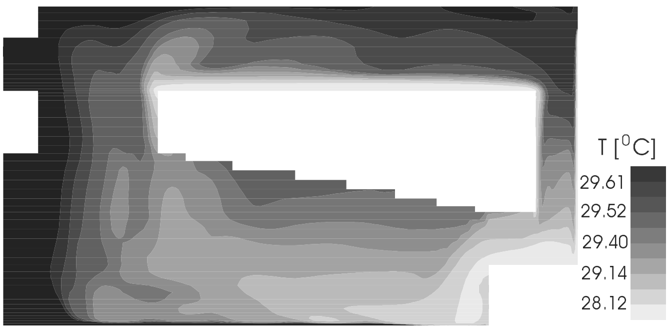


Figure 7: Calculated temperature field in the mid plane of the computational domain.

In order to find reasons for this deviation one should compare the calculated flow field to the experimental one. The calculated flow field for SUCOS-3D is very similar to the experimentally found and calculated one for SUCOS-2D. A stable natural convection loop develops, Fig. 8: the heated fluid rises from the copper plate through the chimney to the covered water level; here the warm flow turns right to the horizontal side area and flows on without an intensive contact to the horizontal cooler; the water is mainly cooled in the vertical side area from where it returns to the pool where it is heated again; part of the cold water from the horizontal coolers moves from time to time in form of cold plumes against the mean flow downwards through the chimney and mixes with the rising heated water. These non-stationary plumes cause the strong time dependence of the heat flux on the copper plate, Fig 5. According to this flow field, the temperatures in the horizontal side area of SUCOS-2D are higher than the ones in the pool under the tilted roof, similar to the temperatures in Fig. 7.

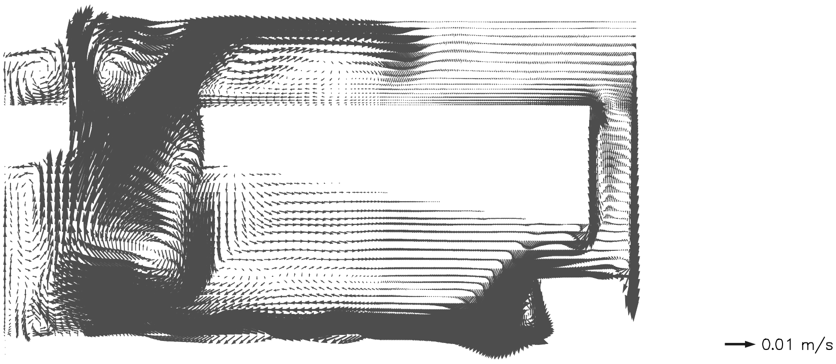


Figure 8: Velocity vector field in the mid plane of the computational domain. Calculation for SUCOS-3D.

The flow field in the SUCOS-3D experiment must be reconstructed from the measured temperatures because no velocity measurements were performed. Different to SUCOS-2D here we find in the experiment the highest temperatures not in the horizontal side area, but below the tilted roof, Fig. 9. Therefore, a different behavior of the natural convection has to be deduced: we have at least to expect stronger mixing between cold counter-current downward flow with the hot rising fluid in the chimney.

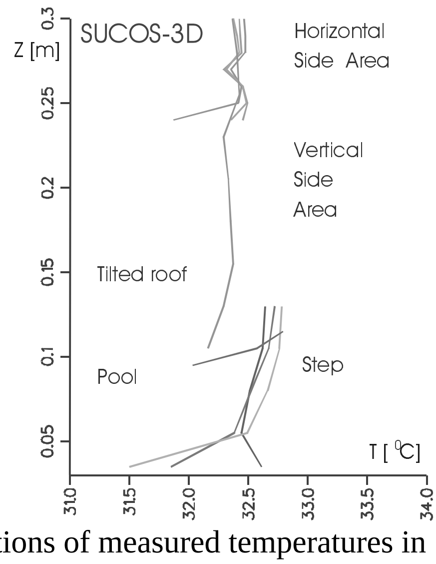


Figure 9: Vertical distributions of measured temperatures in SUCOS-3D (Knebel 1999).

Several reasons for the disagreement in the pool temperature and in the natural convection loop were investigated. Since SUCOS-2D calculations showed a high sensitivity of the natural convection on small thermal disturbances, the thermocouple support structure installed in the chimney was additionally modeled as a thermally interacting structure, but the temperature results in the pool were only slightly improved. A further numerical study was performed changing the kind of the thermal boundary conditions for the active vertical coolers: values for the wall heat fluxes deduced from the experiment were pre-set instead of using surface temperatures. Then, the calculated pool temperature $T_{p,cal}$= 32.8 ^{0}C agrees well with the experimental one, Fig. 10: the deviation from the experiment is reduced from 34% to only 2%. Despite of this positive result, qualitatively the same natural convection loop is obtained like in the previous calculations. This means, the calculated flow field still shows no agreement with the reconstructed one for the experiment.

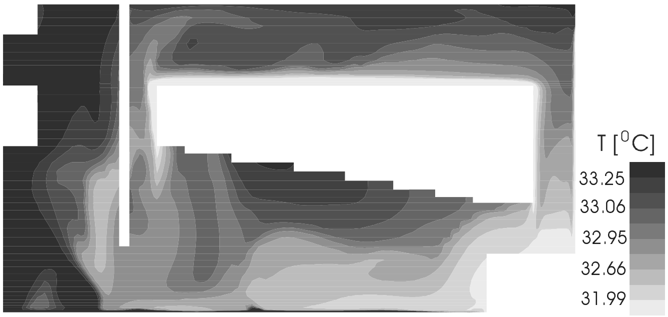


Figure 10: Temperature field in the mid plane of the computational domain. Calculation with temperature probe support and pre-set wall heat fluxes for the vertical coolers.

The differences between experiment and calculation in the behavior of the flow field were further analyzed by means of vertical temperature distributions, which were measured in the chimney and in the vertical side area (Carteciano et al. 2000). The origin of the cold water under the roof in the experiment was reconstructed by analyzing the cooling performance of the horizontal coolers which are divided into two big and two small ones. The cooling performance of one small cooler is in the experiment as high as the one of a big cooler despite the cooling surface ratio of about 1:2. Therefore, a stronger water flow was obviously present over the small cooler. This cold flow returns to the corners of the chimney (Fig. 11) because cold fluid is recorded by the thermocouples at position R below the chimney.

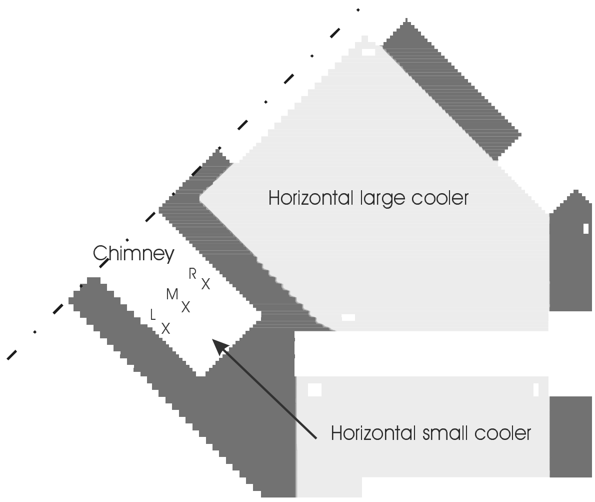


Figure 11: Top view on the numerically modeled geometry with the horizontal coolers. The arrow indicates the increased cold flow coming back to the chimney. The small holes indicate the positions of the feed pipes to the coolers.

As a reason for the increased back flow in the chimney we postulate that one of the horizontal coolers was slightly tilted against the main flow direction. Additional calculations with a 2d slab model similar to SUCOS-2D were performed with different inclinations of the horizontal coolers from 0 to 4 mm, corresponding respectively to 0 and 1.04 % slope. The mass flow of the cold water going back to the chimney from the horizontal coolers increases due to this measure by about 70% and the corresponding heat removal by this flow increases by about 55%! Therefore, the mixing between the cold water and the heated water rising through the chimney is strongly increased like in the experiment. These results show that a very small slope of the horizontal coolers can influence the flow behavior in a drastic way. This would explain the experimental flow field, but a final verification is not possible because the experimental facility is already disassembled.

4. Improvement of the cooling concept

Technical solutions for safety related problems should not be as sensitive against any physical or geometrical parameters as it was detected in the interpretation of this experiment. Here the sensitivity is mainly caused by the lack of stabilizing phenomena which are in natural convection usually the friction forces; these are in this sump cooling concept of negligible magnitude. Thus one should look for other measures to stabilize and increase the flow at least along the horizontal coolers.

It was found above that a small inclination of a horizontal cooler strongly increases the backflow in the chimney. Thus, a positive measure to increase the mass flow in the whole system follows straight forward: the horizontal cooler is inclined by a certain angle in the flow direction; here about 2° are chosen as an example for a small angle, see Fig. 12. The recent calculation is performed for the simpler SUCOS-2D geometry in the general curvilinear coordinate system (Jin 2001). The modification in the geometry leads to an increase in the velocities in the boundary layer above the horizontal cooler and thus also in the vertical side area.

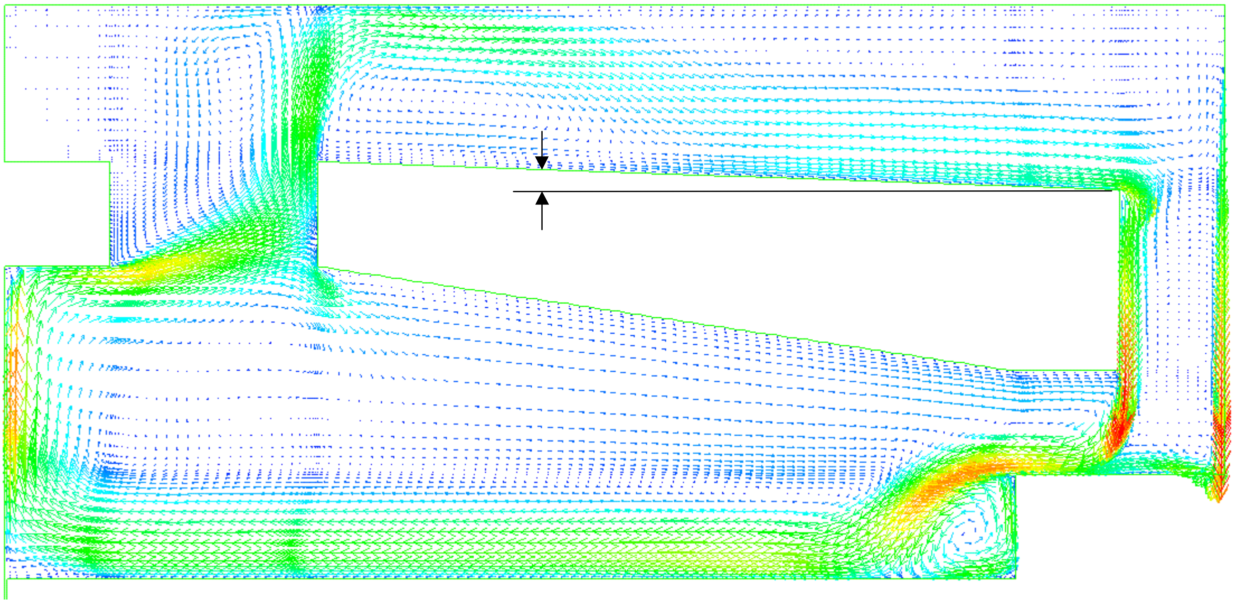
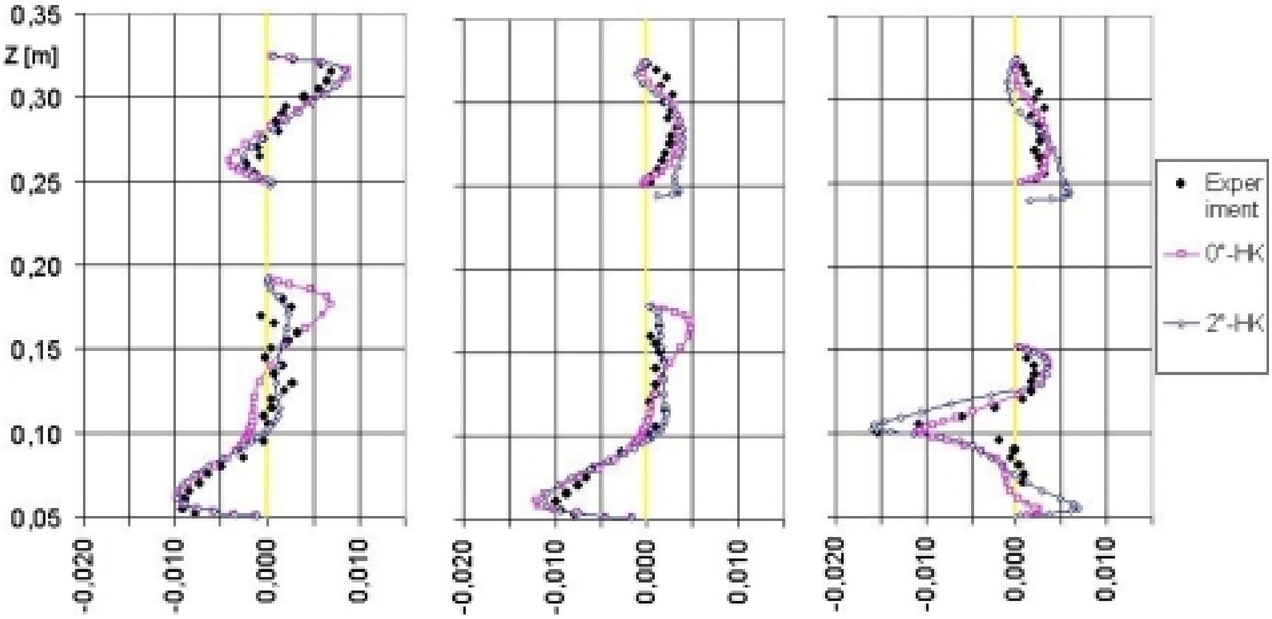


Figure 12: Calculated velocity field in the improved SUCOS-2D concept. Velocity maximum = 0.0187 m/s.

The detailed analysis of the vertical profiles for the time averaged horizontal velocity shows, the counter clock-wise back flow into the chimney is reduced in the tilted channel and the clock-wise flow directly above the tilted horizontal cooler is intensified, Fig. 13. The increase in the velocities forms a stronger jet flow over the step; consequently a larger recirculation area is formed below this jet. In total, the mass flow rate in the chimney grew by about 24%. Thus, indeed an improved cooling capability for the core melt is achieved in combination with a channel geometry which is less sensitive against minor geometric distortions. On the other hand, the maximum temperature in the test rig is not reduced by that amount which corresponds to the increase in the mass flow rate; it is only decreased by 3.4%. Some of the main reasons for this unexpected small effect are due to the decrease of the mixing of hot and cold fluid in the chimney, and due to the increased extension of the recirculation zone below the jet over the step, which reduces the cooling rate of the copper plate in this area.



a) b) c)

Figure 13: Measured and calculated vertical velocity profiles in m/s in the pool and above the horizontal (0°-HK) and tilted (2°-HK) cooler for SUCOS-2D. The profiles are for a) x=0.22m, b) x=0.33m, c) x=0.47m.

5. Conclusions

The former numerical interpretation of SUCOS-2D experiments with the FLUTAN computer code showed that good agreement between experiment and calculation can be achieved when local thermal disturbances like the feed water pipes to the coolers are recorded in the simulation. Here, the single phase natural convection experiment SUCOS-3D is interpreted. Despite recording all important phenomena in the simulation, some discrepancies are found by analyzing the experimental results from SUCOS-3D: in these experiments the maximum temperature did not occur far above the horizontal coolers as in SUCOS-2D, but below the tilted roof, and the heat fluxes over the horizontal and vertical coolers were not homogeneously distributed. Comparable pool temperatures could only be achieved numerically by using the measured heat fluxes at the coolers instead of temperature boundary conditions, but the calculated flow pattern was still different. One explanation for this strange result is supported by the experimental data, which indicate a larger backflow in the chimney in the 3d experiment. This can be explained by postulating that one of the horizontal coolers was slightly tilted against the main flow direction. Additional numerical investigations indeed show that a slope of only one percent would explain the experimental flow field. From this problem of the experiment one can learn how to improve this sump cooling concept: foreseeing a small slope of the horizontal coolers downwards in the expected flow direction stabilizes the flow and increases considerably the efficiency of the horizontal coolers. This result was numerically confirmed by applying the newly developed discretization method using body-fitted coordinates, staggered grids, and a general coordinate transformation. All these numerical results confirm, natural convection in large pools, in which pressure drops are negligible, is very sensitive against small disturbances.

Acknowledgements

This work was financially supported by a contract between German Utilities together with Siemens and Forschungszentrum Karlsruhe. X.J. honors the funding by Siemens-KWU of the work on the development of the numerical scheme for non-orthogonal body-fitted coordinates.

References

Alsmeyer, H., 2000. *Proceedings of the OECD workshop on ex-vessel debris coolability*. Karlsruhe November 15-18, 1999, FZKA 6475.

Alsmeyer, H., Tromm, W., 1999. *The COMET concept for the cooling of core melts: Evaluation of the experimental studies and use in the EPR.* FZKA 6186, EXV-CSC(99)-DO36.

Baumann, W., Carteciano, L., Weinberg, D., 1997. *Thermal propagation effects in a vertical turbulent flow behind a jet block - A benchmark exercise.* Journal of Hydraulics Research, Vol. 35, pp. 843-864

Bonhomme, N., Krugmann, U., 1993. *Safety and containment of the European Pressurized Reactor (EPR).* 5th Int. Seminar on Containments of Nuclear Reactors, Karlsruhe, Germany.

Borgwaldt, H., 1990. *CRESOR, A Robust Vectorized Poisson Solver Implemented in the COMMIX-2(V) Thermal-Hydraulic Code.* Paper AC064. Int. Conf. On Supercomputing in Nuclear Applications (SNA 90). Mito City, Japan, March 12-16, pp. 346-351.

Carteciano L.N., Weinberg D., Müller U., 1997. *Development and Analysis of a turbulence Model for Buoyant Flows.* Proc. of the 4th World Conf. on Experimental Heat Transfer, Fluid Mechanics and Thermodynamics, Bruxelles, Belgium, June 2-6. Vol. 3, S. 1339-46. Pisa Edition ETS.

Carteciano, L.N., Dorr, B., Grötzbach, G., 1999. *Theoretical Interpretation of the Single Phase Sump Cooling Experiment SUCOS with the FLUTAN Code*. Proc. 9th Int. Meeting on Nuclear Thermal-Hydraulics NURETH-9, CD-Rom Log 107.

Carteciano, L. N., Wörner, M., Grötzbach, G., 1999b. *Erweiterte Turbulenzmodelle für technische Anwendungen von FLUTAN auf Naturkonvektion.* Jahrestagung Kerntechnik 1999, Karlsruhe, 18. - 20.5.1999, INFORUM Bonn, S. 129 - 133

Carteciano, L.N., Dorr, B., Grötzbach, G., 2000. *Numerical Investigation of the Single Phase Natural Convection in Sump Cooling Experiments with the FLUTAN Code.* The Tenth Int. Conf. On Emerging Nuclear Energy Systems (ICENES 2000), Petten, The Netherlands, 24.-28. Sept.

Grötzbach, G., Wörner, M., 1999. *Direct numerical and large eddy simulations in nuclear applications.* Int. J. Heat and Fluid Flow (20), pp. 222 - 240

Grötzbach, G., M. Wörner, T. Ammann, A. Blahak, L. Carteciano, B. Dorr, Y. Kimhi, W. Olbrich, W. Sabisch, Q. Ye, M. Alef, St. Genz, A. Hennemuth, G. Janßen, M. Linder, D. Seldner, 1997. *Entwicklung von Thermofluiddynamikprogrammen und ingenieurtechnische Anwendungen,* Projekt Nukleare Sicherheitsforschung, Jahresbericht 1996, FZKA 5963, pp. 411 - 427

Grötzbach, G., Carteciano, L.N., Dorr, B., 2000. *Numerical Analysis of Experiments modelling LWR Sump Cooling by Natural Convection.*. International Atomic Energy Agency, Technical Committee Meeting on “Natural Circulation Data and Methods for Innovative Nuclear Power Plant Design”, Vienna, Austria, July 18-21, TEC-DOC to appear

Günther, C., 1992. *Fortgeschrittene Upwind-Differenzen-Verfahren zur numerischen Lösung der Konvektions-Diffusionsgleichung*. Habilitation, Universität Karlsruhe, KfK 4697.

Hanjalic, K., 1994. *Achievements and limitations in modelling and computation of buoyant turbulent flows and heat transfer*. 10th Int. Heat Transfer Conf., August 14-18, Brighton, U.K.

Hanjalic, K., 1999. *Second-Moment Closures for CFD: Needs and Prospects*. Int. J. Comp. Fluid Dyn. 12, 67-97

Jin, X., 2001. *Rechenverfahren zur Diskretisierung von Strömungen in komplexer Geometrie mittels körperangepasster Gitter.* Dissertation, Universität Karlsruhe. FZKA 6596

Knebel, J. U., 1999. Personal communication.

Knebel, J. U., Müller, U., 1995. *Experimental and Numerical Simulation of Passive Decay Heat Removal by Sump Cooling After Core Melt Down.* Proc. 8th Int. Meeting on Nuclear Thermal-Hydraulics NURETH-8, Vol. 2, pp. 1059-1066.

Knebel, J. U., Müller, U., 1997. *Scaling of passive decay heat removal by sump cooling.* Jahrestagung Kerntechnik 97, Aachen, 13.-15. Mai, INFORUM Bonn, S.144-47

Kuczera, B., Eglin, W., Weisshäupl, H.A., 1994. Towards an enhanced quality in pressurized water reactor safety. Kerntechnik 59, pp. 151 – 155.

Kuhn D., 1996. *Numerische und experimentelle Untersuchungen zum Sumpfkühlkonzept*. Diplomarbeit, Institut für Angewandte Thermo- und Fluiddynamik (IATF), Forschungszentrum Karlsruhe.

Leonard, B. P., 1979. *A Stable and Accurate Convective Modelling Procedure Based on Quadratic Upstream Interpolation.* Computer Methods in Applied Mechanics & Engineering, Vol. 19, p. 59.

Weinberg, D., Rust, K., Hoffmann, H., 1996. *Overview Report of RAMONA-NEPTUN Program on Passive Decay Heat Removal*. FZKA 5667.

Weisshäupl, H.A., Bittermann, D., 1995. *Large spreading of core melt for melt retention–stabili-zation.* Nucl. Engng. & Design 157, pp. 447 – 454.

Willerding, G., Baumann, W., 1996. *FLUTAN 2.0 input specifications.* FZKA 5712.

Wörner, M., Grötzbach, G., 1997. *DNS database of turbulent natural convection in horizontal fluid layers.* World Wide Web-site http://www.fzk.de/irs/turbit