Numerical analysis of experiments modeling LWR sump cooling by natural convection

G. Grötzbach, L.N. Carteciano, B. Dorr
Forschungszentrum Karlsruhe GmbH, Institut für Reaktorsicherheit, Germany

Abstract. An optional sump cooling concept for the European pressurized water reactor EPR was investigated at the Research Center Karlsruhe. This concept foresees to utilize single phase natural convection in water to remove the decay heat from the core melt. The natural convection was investigated by the SUCOS-2D and -3D scaled experiments. A numerical investigation and interpretation of these experiments was performed by means of the computer code FLUTAN. In this paper, the numerical investigation of SUCOS-3D is summarized. Following the results of the former 2d experiments and the numerical analysis of both experiments, an unexpected temperature distribution is found in this 3d experiment. Basing on the experimental data it had to be postulated that one of the horizontal coolers was slightly tilted against the main flow direction. Additional numerical investigations show that a slope of only one percent would explain the experimental flow field. Conclusions are also drawn on the limits of scalability and transferability of the experimental results to a reactor sump. A detailed transformation will only be possible by applying well validated CFD-codes and experienced code users. As the flow in the reactor sump will be turbulent and this flow is strongly three-dimensional and time-dependent, only the method of Large Eddy Simulation is considered of being an adequate tool for reliable transformation of the gained experience to analyses for the reactor sump at 1:1 scales.

1. INTRODUCTION

The final safety barrier after a core melt down accident is the core catcher in the reactor sump. An optional cooling concept for the European Pressurized water Reactor EPR 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 (see Fig. 1) is foreseen. Then cooling of the core melt begins with the water from the in-containment refueling water storage tank. Water cooled heat exchangers and condensers are present in the reactor sump region in order to remove the decay heat from inside the containment. The decay heat is transferred from the core melt to the sump water by evaporation, natural convection, and conduction. In the first days the convection of the sump water is in two-phase conditions; about ten days after the accident, single-phase natural circulation conditions are reached.

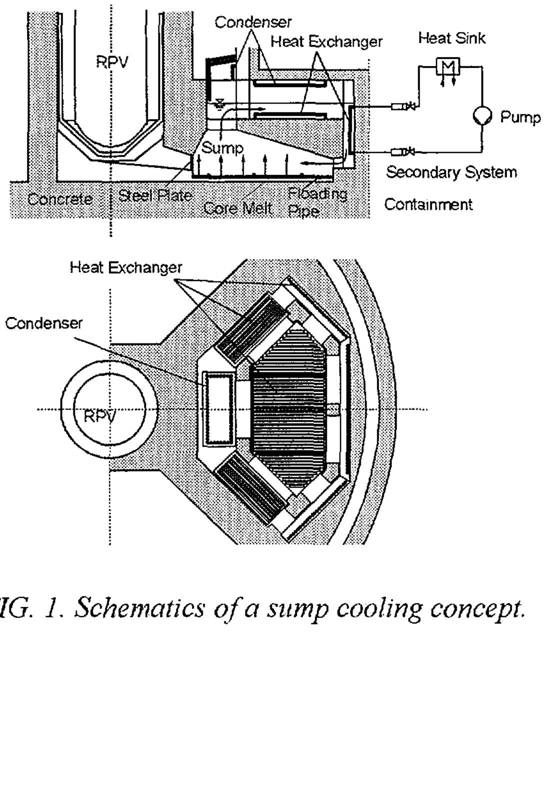

FIG. 1. Schematics of a sump cooling concept.

The single phase natural convection is experimentally and numerically investigated in the Research Center Karlsruhe in the program SUCOS (SUmp COoling Small) (Knebel et al. 1995). The aim of this program is to obtain quantitative results to be transferred to the prototypic condition in order to make a statement on the feasibility of the single phase sump cooling. Two scaled test facilities (1:20) are 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 (1298 × 580 ×= 275mm) of the sump. Water was heated by a heated 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.

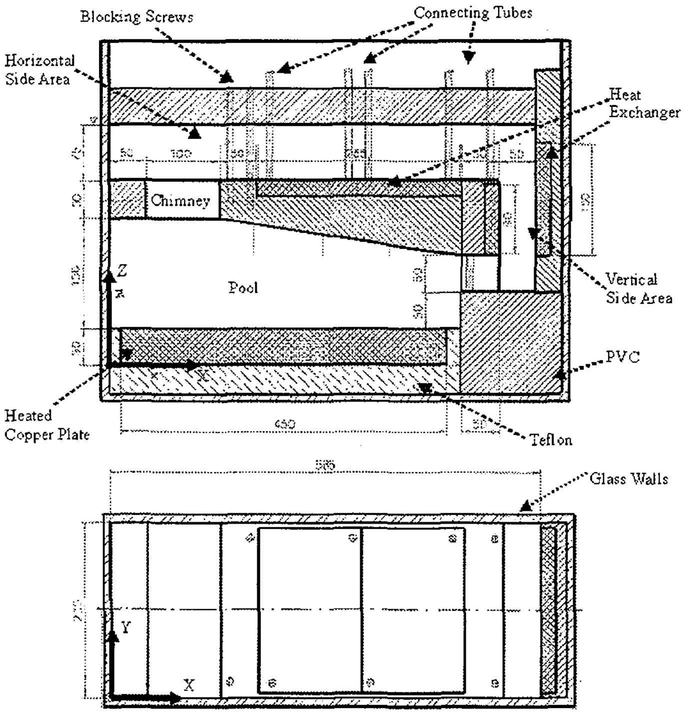

FIG. 2. Schematics of the test geometry SUCOS-2D.

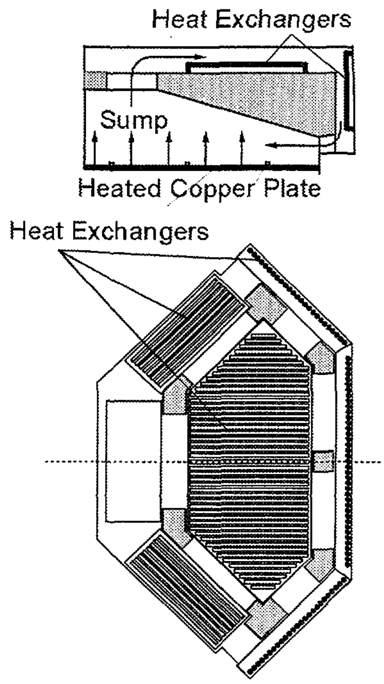

FIG. 3. Schematics of the test geometry SUCOS-3D.

A numerical investigation of this sump cooling concept and the related model experiments is performed by using the FLUTAN code (Willerding at al. 1995). This thermal- and fluid-dynamical computer code is developed in the Research Center Karlsruhe for the numerical analysis of the passive heat decay 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 confirm the feasibility of the sump cooling concept and to analyze in more detail the experiments.

The first step of this investigation consisted of the numerical simulation and interpretation of an SUCOS-2D experiment (Carteciano et al. 1999). The most important result was that SUCOS-2D cannot be well reproduced with a two-dimensional calculation whereas three-dimensional calculations reproduce the experiment quite well. The simulations showed that significant three-dimensional experiment specific phenomena are present in the experiments, but these 3d phenomena are of much less relevance in the reactor sump. Furthermore, the analysis of the calculation of SUCOS-2D gave information on the requirements for the modeling of geometry and boundary conditions for simulations of an experiment of SUCOS-3D which is the second and final step of the numerical investigation of the single-phase sump cooling concept. This step of analyzing the SUCOS-3D experiment in detail is documented in (Carteciano et al. 2000). Here, the most important results of this study are discussed to give at the end an outlook on the current status of methods to transfer all these results of the model experiments by CFD tools to EPR scales.

2. FLUTAN CODE

FLUTAN is a highly vectorized computer code for 3D fluiddynamic and thermal-hydraulic analyses in Cartesian or cylinder coordinates. It is related to the family of COMMIX codes originally developed at Argonne National Laboratory, USA (Shah et al, 1985). FLUTAN was developed in order to simulate single phase flows with small compressibility. The conservation equations for mass, momentum, energy, and turbulence quantities are discretized in a structured grid 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 at least a powerful tool 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.

3. SUCOS-3D INVESTIGATION

3.1 The test facility

The test facility SUCOS-3D consists of a tank (1298 × 580 × 275) whose outer walls are 20 mm thick and made of Plexiglas (Fig. 3). The ratio between lengths in the test facility and EPR is 1:20. A 30 mm thick copper plate which is heated by electric heat 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, 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 perform 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 20^{0}C and the flow rate was 20 or 40 g/s. The measured pool temperature was 32.6 ^{0}C.

3.2 Specification of the computational model

To reduce the computational effort only half of the complex but symmetric geometry of the test facility is simulated. 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 surface temperature or of 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. A linear distribution for the 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 inlet and outlet coolant water is less than 1 K. The prescribed values are determined by means of experimental data.

In former simulations for SUCOS-2D it was found that the heated copper plate needs special attention (Kuhn 1996). Even the developing circulation sense in the complete 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. 4. 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.

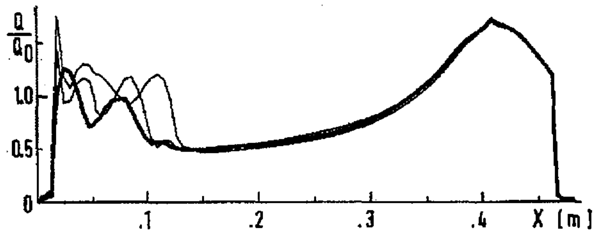

FIG. 4. Horizontal distributions of the calculated heat flux Q divided by its mean value Q_0 from the copper plate to the fluid at three different times in SUCOS-2D.

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

All following results are from the transient calculation and are time averaged over two minutes, like in the experiment. The calculated temperature field is shown in Fig. 5. 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 corresponding deviation is $(T_{p,cal}-T_{p,exp})/(T_{p,exp}-T_{cool})$ = 34%.

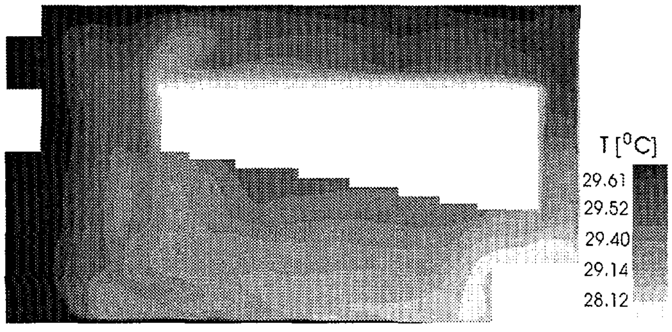

FIG. 5. Calculated temperature field roughly in the mid plane of the computational domain.

In order to find possible 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. 6: 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 4. 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. 5.

The flow field in the experiment SUCOS-3D must be reconstructed from the measured temperatures because no velocity measurements were performed. Other than in SUCOS-2D here we find in the experiment the highest temperatures not in the horizontal side area, but below the tilted roof, Fig. 7. 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 hot rising fluid in the chimney.

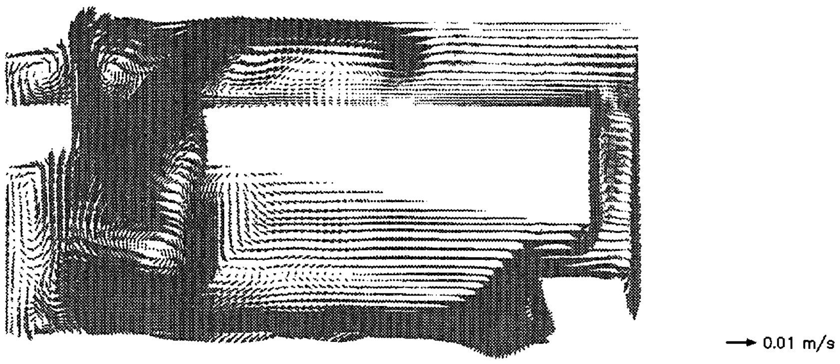

FIG. 6. Velocity vector field roughly in the mid plane of the computational domain. Calculation of SUCOS-3D.

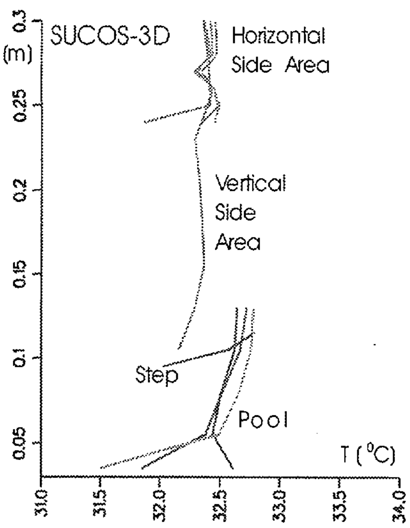

FIG. 7. Distributions of measured temperatures in SUCOS-3D.

Possible 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 thermal disturbances, the thermocouple support structure installed in the chimney was additionally modeled as a thermally interacting structure. Unfortunately, the exact position of the movable probe is not known. Nevertheless, the new results with probe support structure are better than the previous ones and bring the calculated pool temperature to the right direction but not yet enough to achieve a satisfactory agreement.

So far, the cooling performance of the vertical coolers was overestimated by all previous calculations. Therefore, 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 $T_{p,cal} = 32.8\ ^0C$ agrees well with the experimental one, Fig. 8: the deviation 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 in the experiment.

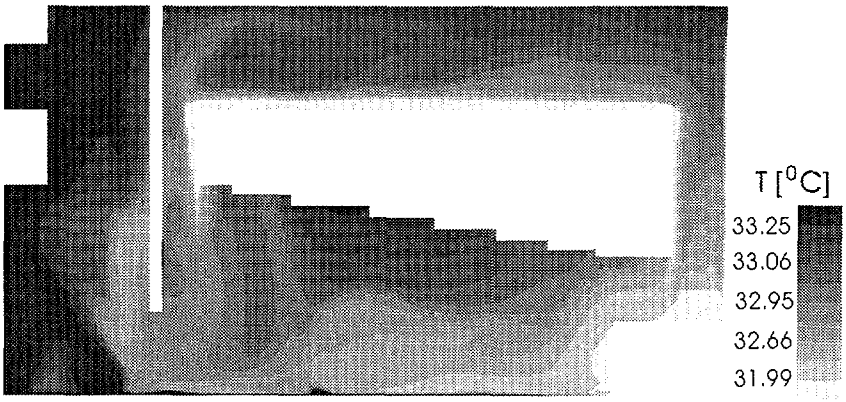

FIG. 8. Temperature field roughly in the mid plane of the computational domain. Calculation with temperature probe support and pre-set wall heat fluxes for the vertical right coolers.

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 amount of water flowing down from the horizontal cooler to the vertical one was much less in the experiment than calculated, which causes reduced heat fluxes at the outermost section of the vertical coolers. An other significant difference between calculation and experiment can also be found in the chimney: recirculation of cold water flowing back from the horizontal cooler to the chimney is calculated, while cold water was registered in the experiment in the plane of measurements only under the tilted roof. 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. 9) and is recorded only when it reaches the thermocouples at position R below the chimney.

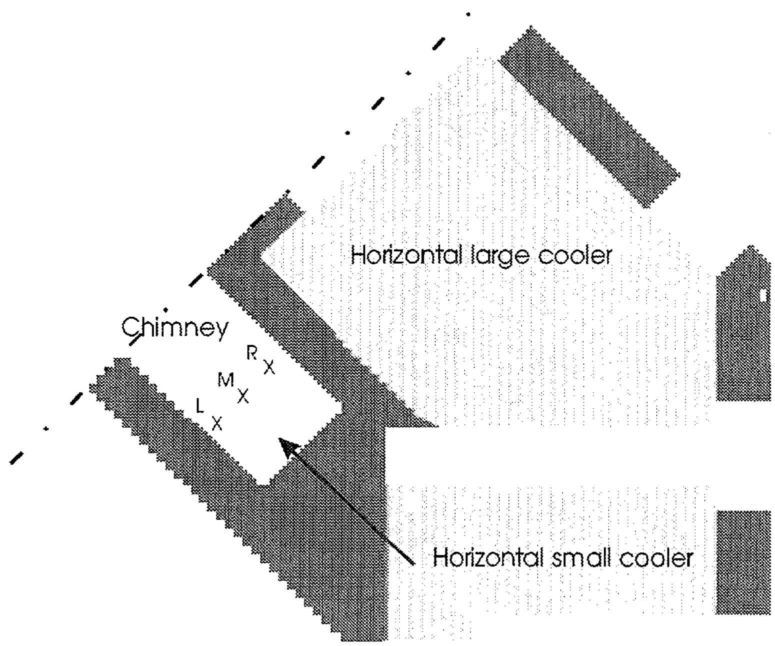

FIG. 9. Arrangement of the horizontal coolers. The arrow indicates the increased cold flow coming back to the chimney.

A decisive conclusion on the disturbance responsible for the experimental flow behavior was not found in all previous calculations and studies. Further deviations in the modeling or boundary conditions used in the simulations from those in the experiments have to be considered. One example, which would explain the unexpected results from the SUCOS-3D experiment, is to 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 and would explain the experimental flow field, but a final check is not possible because the experimental facility is already disassembled.

4. TRANSFER METHODS TO EPR CONDITIONS

Conclusions on the conditions in the reactor sump can in principal be drawn from the experimental results by two means, by applying scaling laws, and by applying CFD methods. Scaling laws were applied by Knebel & Müller (1997) to determine from the 2d experiments that in the first days the temperatures in the water above the core melt will be large enough to achieve boiling. As a consequence, the investigations in SUCOS are only applicable for the long term cooling after about ten days. A condition for applying scaling laws is, that the physical transport phenomena are of similar relevance in the model experiment and in the reactor sump. The analyses of SUCOS-2D in Carteciano et al. (1999) have shown that specific 3d effects, like the feed water pipes to the coolers, the bolts, and screws going through the fluid domain strongly affect the experimental results. In the small cross section of the SUCOS-2D slab geometry these structures become more important for the heat transfer and for the flow resistance than in the huge reactor sump. Thus, the results of this experiment cannot be used for scaling up to predict accurately the temperatures in the reactor sump. In contrast, the first numerical cases, which did not record those structures, are more feasible as a basis for this extrapolation. The experimental results of SUCOS-3D could be a better basis because there the cross sections occupied by those structures are of relatively less importance, but there the above mentioned strange flow distribution was found which is specific for this experiment and not for the reactor sump.

Scaling up by CFD-tools is also not free of serious problems in this context. On one hand the SUCOS tests could successfully be interpreted with the FLUTAN code. Such CFD codes have the flexibility to tackle all the experiment or reactor specific structures. On the other hand the model experiments showed laminar flows, whereas the reactor sump will have turbulent flow conditions (Knebel & Müller 1997). Hence additional investigations are necessary by experiments of similar flow types to validate the turbulence models and boundary conditions used with the turbulence models. Purely buoyant flows are currently 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). An additional feature, which is inherent to most purely buoyant flows, is its local time dependence. The cold plumes plunging down through the chimney are a low frequent phenomenon which was also not completely filtered out in the experiment by time-averaging over two minutes. This causes the wall heat flux on the copper plate to change in time, Fig. 4. And also the hot plumes rising from the surface of the copper plate are no stationary phenomenon. The surface temperature on the copper plate, Fig. 10, is very straggly as it is typically found in Rayleigh-Bénard convection in which the hot plumes rise mainly from the knots of those straggly structures (Wörner & Grötzbach 1997).

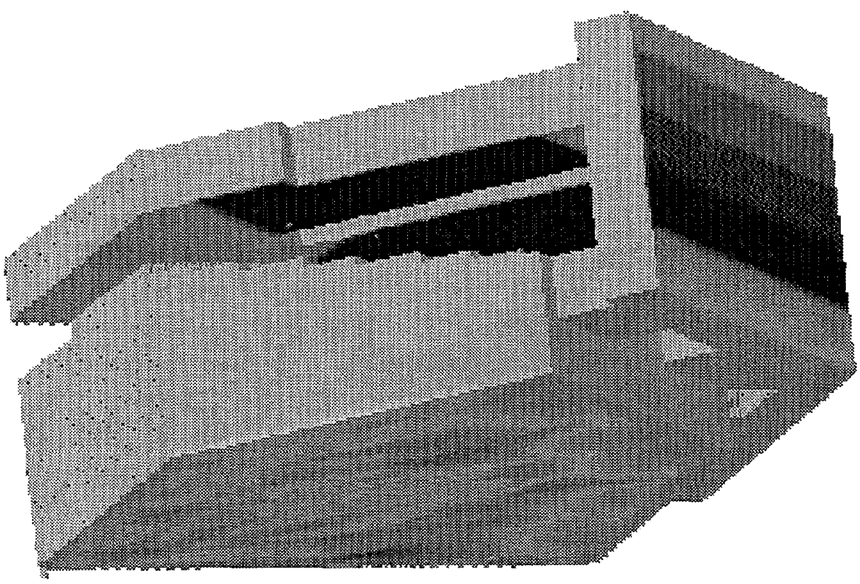

FIG. 10. Calculated temperature field on the surface of the structures of SUCOS-3D. View from below on the fluid domain. The dark areas represent the cooler surfaces.

As a consequence of the 3d and time-dependent nature of buoyant flows and of the current status of development of the standard turbulence models, there is currently no way to achieve reliable results by standard models on the temperature fields in the reactor sump. 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; for an overview see Grötzbach & Wörner (1999). These show the tremendous potential of the LES method and that its possibilities are going far beyond those of standard Reynolds averaged turbulence models. The main problems which need to be solved for LES methods in this context are e.g. the development of more universal subgrid scale models, of boundary conditions for buoyant flows, and of numerical methods in commercial codes that fulfill LES requirements.

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. The numerical results confirm, natural convection in large pools, in which pressure drops are negligible, is very sensitive against small disturbances. Here, some other 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, this is to postulate 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 would stabilize the flow and would drastically increase the efficiency of the horizontal coolers.

Based on the performed numerical investigations it can be concluded that a transformation of the experimental results from SUCOS-2D and -3D to reactor sump conditions by means of scaling laws is questionable. Such transformation will only be possible by applying well validated CFD-Codes and experienced code users with a sound physical and engineering background. Current standard turbulence models form the working basis for engineers, but they can only be used for approximate predictions, because the statistical models fail for buoyant flows which are locally time-dependent. The only more reliable solution could come from adequate Large Eddy Simulation methods and LES-suitable codes for complex geometries.

ACKNOWLEDGEMENTS

This work was partly financially supported by a contract between German Utilities together with Siemens and Forschungszentrum Karlsruhe.

REFERENCES

[1] 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.

[2] 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 Application (SNA 90). Mito City, Japan, March 12-16, pp. 346-351.

[3] 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.

[4] 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 Log107.

[5] 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.

[6] 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.

[7] GRÖTZBACH, G., WÖRNER, M., 1999. *Direct numerical and large eddy simulations in nuclear applications.* Int. J. Heat and Fluid Flow (20), pp. 222-240.

[8] 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.

[9] GÜNTHER, C., 1992. *Fortgeschrittene Upwind-Differenzen-Verfahren zur numerischen Lösung der Konvektions-Diffusionsgleichung.* Habilitation, KfK 4697.

[10] 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.

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

[12] 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.

[13] KNEBEL, J. U., MÜLLER, U., 1997. *Scaling of passive decay heat removal by sump cool ing.* Jahrestagung Kerntechnik 97, Aachen, 13.-15. Mai, INFORUM Bonn, S.144-47.

[14] KUHN D., 1996. *Numerische und experimentelle Untersuchungen zum Sumpkühlkonzept.* Diplomarbeit, Institut für Angewandte Thermo- und Fluiddynamik (IATF), Forschungszentrum Karlsruhe.

[15] 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.

[16] SHAH et al., 1985. *COMMIX 1-B: A 3d Transient Single-Phase Computer Program for Thermal Hydraulic Analysis of Single and Multicomponent Systems.* NUREG/CR-4348, Vol. I and II.

[17] WEINBERG, D., RUST, K., HOFFMANN, H., 1996. *Overview Report of RAMONA-NEPTUN Program on Passive Decay Heat Removal.* FZKA 5667.

[18] WILLERDING, G., BAUMANN, W., 1996. *FLUTAN 2.0 input specifications.* FZKA 5712.

[19] 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.