Proc. IX ENFIR = IX Brazilian Meeting on Reactor Physics and Thermal-Hydraulics, October 25-29, 1993, □
Caxambu, Minas Gerais, Brazil, □
Comissao Nacional de Energia Nuclear, Centro de Desenvolvimento da Energia Nuclear, Belo Horizonte, □
CDU 621.039.51, 532.5, 1993, pp. 258 - 264

DIRECT NUMERICAL SIMULATION OF TURBULENCE IN NUCLEAR THERMAL-HYDRAULICS

G. Grötzbach, M. Wörner
Kernforschungszentrum Karlsruhe
Institut für Reaktorsicherheit
Postfach 3640, D-76021 Karlsruhe
Germany
Tel. 07247/824477

ABSTRACT

Direct numerical and large eddy simulation methods are universal and reliable approaches for studying mechanisms of turbulent flows. They need sufficiently fine discretizations or adequate models for small scale effects. The principles of the methods are discussed and an overview on some reactor related applications is given. The first contributions were by the large eddy simulation method and appeared roughly two decades ago. A few years later the first contributions by the direct simulation method appeared. All earlier applications are for simple flows, whereas just recently applications to more realistic technical flows appear. As an example for new classical direct numerical simulations results are presented for turbulent Rayleigh-Bénard convection in liquid sodium and air at similar Grashof numbers. Terms in the transport equations of turbulent heat flux are evaluated from the numerical results, using their analytical definitions. It is found that for liquid metals those model assumptions for turbulence terms need adaptation to the influence of molecular Prandtl number which are influenced by the temperature field. The diffusive transport of turbulent heat flux in natural convection is found to be mainly due to the action of pressure fluctuations and not, as usually assumed, due to velocity fluctuations. The corresponding conventional model assumption for turbulent diffusion is analysed.

INTRODUCTION

Most single phase flow problems, which have to be considered in nuclear reactor technique, are in the turbulent flow regime. This means, these flows are locally three-dimensional, time-dependent and mostly irregular. The conservation equations for any type of flow are known. Thus, full turbulence simulations should be possible. The problem of their solution is that turbulent flows have a spectrum of length scales extending from large vortices down to small vortices, which all are physically important and therefore have to be resolved numerically. With increasing turbulence level the width of the spectrum increases. Thus, currently direct numerical simulations (DNS) are only feasible for small turbulence levels. At larger turbulence levels the small scales which are not resolved by the grid have to be modelled by subgrid scale models; this type of simulation is called large eddy simulation.

For practical problems engineers use statistical models in which turbulence has completely been replaced by physical models. In reactor applications several types of flows have to be considered which need special treatments and adaptations of these models. Natural convection in closed containers with stable and unstable thermal stratification and with fluids of different Prandtl numbers is an example. Especially the turbulence in natural convection of liquid metals is of relevance in safety analysis of fast breeder reactors. For this field many scaled model experiments are performed using water as a model fluid [1,2].

The interpretation of the experiments and the transfer of the results to reactor conditions, where liquid sodium is used as coolant, is mainly done by computer codes [3,4]. Turbulence models used in these codes are being improved to be applicable to purely buoyant flows, especially in liquid metals [5]. Thus detailed turbulence data of such flows are required to calibrate turbulence models and to improve model assumptions for this special application. Due to difficulties in working with liquid metals only very few experimental data suitable for model validation are available. Here we use the method of direct numerical simulation of turbulence to provide such data for at least small turbulence levels.

In this paper we give an introduction to the simulation method and the computer code TURBIT [6]. We discuss special features of liquid metals and the consequences for an efficient numerical scheme, as there is the need for semi-implicit time integration of the thermal energy equation. Then we give a short overview on contributions of direct and large eddy simulations to fundamental problems of nuclear reactor thermal hydraulics. The body of the paper is on new results for Rayleigh-Bénard convection, that is the convection in an infinite horizontal fluid layer heated at the lower and cooled at the upper wall. We use results of our numerical simulations to analyse macroscopic mechanisms and structures of the flow and statistical features of turbulence in liquid sodium and air. In continuing a previous study [7] where we looked at the concept of turbulent Prandtl number for modelling the turbulent heat flux we now analyse terms of the transport equation of turbulent heat flux. We show that the pressure contribution to the turbulent diffusion in this equation is of special importance in natural convection, but this contribution is neglected in existing codes.

SIMULATION METHOD

The basic equations for laminar and turbulent flow with heat transfer are the conservation equations for mass, momentum, and thermal energy. For a fluid with constant material properties they can be written in terms of u_i and g_i for the components of the velocity and gravity vector respectively (i = 1,2,3), p for pressure, and T for temperature:

$$\nabla_i u_i = 0 \tag{1a}$$

$$\nabla_t u_i + \nabla_j (u_i u_j) = \nabla_j (1/\sqrt{Gr}\ \nabla_j u_i) - \nabla_i p - (T_{ref} - T)\,\delta_{i3} \tag{1b}$$

$$\nabla_t T + \nabla_j (T u_j) = \nabla_j (1/(Pr\sqrt{Gr})\ \nabla_j T) + Q\,. \tag{1c}$$

The symbols used are t for time, Gr for Grashof number $|g| \beta \Delta T_w D3/v2$, β for volumetric expansion coefficient, ΔT_w for difference between wall temperatures, D for channel height, Pr for Prandtl number v/a, v and a for the diffusivities of momentum respectively heat, and T_{ref} for reference temperature.

The method of direct numerical simulation of turbulence is based on the full conservation equations of mass, momentum and thermal energy. The features of turbulence require to solve these equations in three dimensions and in time, and to use grids which resolve the largest and smallest scales of turbulence. These requirements are met in the simulations which we present in this paper. Therefore no assumptions for the subgrid scales and no wall models are needed and the simulations do not depend on any model coefficients.

The large eddy simulation method is also based on the full set of three-dimensional time-dependent conservation equations. The small scales of turbulence which are not resolved by the grids are accounted for by subgrid scale models. These models look like statistical models. The most important differences are that the eddy diffusivities and eddy conductivities are determined from the fluctuating resolved scales and that the length scales depend on the actual grid widths. The influence of these models decreases with decreasing grid width, this means with increasing spatial resolution capabilities of the grid. At large turbulence levels the small scales are roughly isotropic. Therefore, the models are more universal than statistical ones. The values of the model coefficients can be determined theoretically by means of the theory of isotropic turbulence using the Kolmogorov spectrum as the only empirical intput [6]. The wall conditions used with large eddy simulations can be discretized without further approximations at small turbulence levels; at large turbulence levels suitable time-dependent formulations of wall laws have to be used.

The computer code TURBIT is based on a finite volume method and allows for direct and large eddy simulations of turbulent flow in simple channel geometries [6]. The basic equations are solved on a staggered grid in a dimensionless form. For normalization the channel height D, velocity $u_0 = (|g| \beta \Delta T_w D)^{1/2}$, time D/u_0 and temperature difference ΔT_w are used. Spatial discretization is done by second order central finite differences. For time integration of the momentum and thermal energy equation the explicit Euler-Leapfrog scheme is used.

Time integration of liquid metal flows needs special consideration. Due to the high thermal conductivity of liquid metals small-scaled disturbances in the temperature field are compensated rapidly. Thus the thermal diffusion process involves a very small time scale. A fully explicit time integration scheme has to resolve this time scale for reasons of numerical stability. Since these time steps are much smaller than those required to resolve even the highest frequencies of turbulence, fully explicit time integration schemes are very inefficient for numerical simulation of liquid metal flows. Therefore we implemented in TURBIT two semi-implicit time integration schemes for the diffusive terms in the thermal energy equation, the Leapfrog Crank-Nicolson and the Adams-Bashforth Crank-Nicolson scheme respectively [8,9]. Both can be used alternatively and allow for an increase of the time step up to a factor of 50 compared to the fully explicit Euler-Leapfrog scheme, which is still used for the momentum equations.

The set of linear equations resulting from the semi-implicit time integration schemes has to be solved efficiently to achieve a remarkable reduction of the computational costs. The resulting set of equations of our semi-implicit time integration schemes is very similar to that of a discretized Poisson equation, since the diffusive term contains the Laplace operator. In TURBIT we use the projection method of Chorin and solve a Poisson equation with Neumann boundary conditions at walls for the pressure at every time step to fulfil the conservation of mass. The vectorized direct solver is based on the fast Fourier transform and Gaussian elimination [10]. This solver was modified to allow for the solution of the set of linear equations and for Neumann and Dirichlet boundary conditions for the temperature. For verification of the new integration method we successfully recalculated a benchmark of 2D convection of a low Prandtl number fluid in a rectangular box[8].

REACTOR RELATED SIMULATIONS

According to the restricted computational power available in earlier times the method applied firstly was the large eddy simulation method. The objectives of simulations performed for forced convection in simple plane channels were to contribute to the fundamental understanding of turbulent flows, to provide turbulence data which can hardly be measured but which are necessary to improve existing statistical turbulence models needed for reactor engineering calculations, and to analyse more qualitatively prototypical problems in nuclear engineering. Meanwhile, the computational power increased considerably. The turbulent natural convection currently of interest in nuclear engineering can be treated by the direct simulation method. And some applications appear considering somewhat more realistic channel geometries.

One of the first reactor related simulations was on the vibration of fuel pins due to pressure fluctuations in subassemblies with forced convection. This was intensively investigated in the seventies. The local pressure fluctuations are a superposition of fluctuations caused by turbulence in the flow and of oscillations created in pumps, bends, junctions, etc. It is very hard to measure data of purely turbulent pressure fluctuations at walls. Therefore, their statistics were determined by large eddy simulations [11]. It is an advantage of the method to have the time-dependent pressure field available. From this field the resulting mechanical forces on the inner rod of an annulus and their statistics were determined. The calculated azimuthal two-point correlations of turbulent pressure fluctuations show a very small correlation at opposite sides of the inner rod. Thus, in experiments the purely turbulent fluctuations could be separated from the transported ones by measuring pressure signals simultaneously at opposite sides of a rod, correlating the data, and comparing these correlations to the calculated ones. Just recently a similar investigation was published in which pressure induced vibrations are analysed by means of the large eddy simulation method in a more realistic geometry typical for a heat exchanger [12]. Unfortunately, there are currently no details available on the methods used by adapting the engineering code to large eddy simulation, especially on the inlet conditions needed.

The first contribution of the large eddy simulation method to an actual prototypical reactor related problem was on a forced vertical channel flow with strong buoyancy contributions. A procedure was developed for setting initial conditions for the blowdown experiments simulating a loss of coolant accident in the HDR reactor in Kahl, Germany. It was planned to feed hot and cold fluid separately to the lower end of the downcomer to establish realistic thermal stratification. The concern was whether both fluids would mix intensively in this gap as required to reach the intended initial temperature distribution or not. There were no experimental data available for such a flow, and hence no verified turbulence models. Instead of performing an expensive separate experiment to investigate the mixing in a vertical channel with strong buoyancy contributions a large eddy simulation was performed with a coarse grid and less than one hour of CPU-time. The results showed that both fluids will strongly mix within the downcomer [13]. Later on the blowdown experiments in the reactor were performed and the numerical prediction was confirmed.

An other investigation of an actual serious reactor related problem using the large eddy simulation method was published recently [14]. There is currently tremendous experimental and theoretical work going on to analyse the thermal striping phenomenon in detail. For the special problem of impinging hot jets of water and sodium large eddy

simulations were performed. The principal mechanism for generation of large-scale temperature fluctuations were found in the instability at the edge of the jet, which causes entrainment of cold fluid into the jet.

Further reactor oriented simulations were on fundamental modelling aspects of heat transfer in liquid metal forced convection which are difficult to investigate experimentally. There are sufficient integral data about the wall heat fluxes, but nearly no reliable data about the statistics of the turbulent temperature fluctuations within the fluid to adapt local turbulent heat flux models [15]. In numerical simulations there are no real difficulties with these fluids. For purely forced flow the large eddy simulation model for momentum does not depend on the molecular Prandtl number. The temperature field, on the other hand, shows only large scale temperature fluctuations because the large thermal conductivity of these fluids suppresses small scale temperature variations. Thus, for these fluids at large Reynolds numbers our selfadaptive subgrid scale model [6] automatically results in a large eddy simulation for the momentum field, and in a direct or nearly direct simulation for the thermal energy field. This means, the temperature fluctuations do not depend on parameters which are a function of the Prandtl number. With such combined simulations data were provided for the statistics of turbulent temperature fluctuations and eddy conductivities for several Reynold numbers, Prandtl numbers, and various thermal wall conditions [16]. With existent turbulence data Lawn [15] could not verify his theory to predict turbulent Prandtl numbers, but the simulations gave the required data.

The first contribution of the direct simulation method on a reactor related problem was on natural convection in an internally heated horizontal fluid layer. For evaluation of the consequences of core melting the heat transfer in molten reactor material had to be analysed. From experiments many data are available on wall heat fluxes. Few data are available on mean temperature profiles for an internally heated fluid layer bounded by horizontal walls of equal temperature. The first consistent set of turbulence data for this flow in the literature were provided by direct simulations for a channel with small horizontal extensions [6,17,18]. They show that the lower 80% of the layer are stably stratified and about half the height therein is dominated by a counter gradient heat flux. Thus, statistical turbulence models using gradient assumptions are not adequate for this type of flow. For more accurate statistical models additional transport equations for the turbulent heat flux will be required. Turbulence data for calibration of such models were also provided by the older simulations and verified with a new simulation in a channel with large horizontal extension [19]. The first experimental data became available few years ago [20]. They cannot be directly compared because they apply for other flow parameters and for different thermal boundary conditions.

CURRENT CONTRIBUTIONS OF DNS METHOD

The current development of new LMFBR designs aims at totally passive decay heat removal from the core by natural convection. Within this development basic understanding of turbulent convection of liquid metals is of dominant importance. Here we consider the Rayleigh-Bénard convection which is characterized by an upwards directed heat transfer through an infinite plane fluid layer bounded by horizontal rigid walls. Many experimental and theoretical investigations have been published for moderate molecular Prandtl numbers, $Pr \approx 1$. Thus, this flow can serve as a test case for codes and turbulence models. It is also typical for many technical heat transfer problems. As we are interested in the turbulence developing in natural convection of liquid metals, we investigate Rayleigh-Bénard convection at small Prandtl numbers.

The Rayleigh-Bénard problem is characterized by two dimensionless numbers, the Rayleigh number $Ra = Gr \cdot Pr$ and the Prandtl number. Here we consider three simulations (see Table 1), two of liquid sodium ($Pr = 0.006$) and one of air ($Pr = 0.71$). Although Ra and Pr are clearly different in the sodium and air cases, the Grashof numbers are of the same magnitude.

Table 1. Parameter and grid data of the simulations

Pr	Ra	Gr	$X_{1,2}$	Δx_{3w}	$N_{1,2}$	N_3
0.006	6,000	10^6	8.0	0.01	200	31
0.006	24,000	$4 \cdot 10^6$	8.0	0.005	250	39
0.71	630,000	$0.9 \cdot 10^6$	7.92	0.005	200	39

Grids used in direct numerical simulations have to meet several requirements to ensure physical relevance of the results. The grid widths Δx_i ($i = 1,2$ horizontal, $i = 3$ vertical) have to be fine enough to resolve the smallest scales of turbulence and Δx_{3w} to resolve the thin boundary layers. The extensions $X_{1,2}$ of the control volume in the horizontal directions, where periodic boundary conditions are used, have to be large enough to allow for development of large scale structures. The numbers of mesh cells N_i follow directly from the values chosen for X_i and Δx_i, except for the vertical direction in which non-equidistant grids are used. For simulations with air sufficient experience and experimental information are available to specify adequate grids [21]. In contrast, a sensitivity study was necessary for liquid sodium to investigate the influence of Δx_i and especially of X_i on the numerical results [7] because there exists roughly no information about the size of the smallest and largest scales.

The simulation for $Ra = 6{,}000$ was started from zero velocities and a linear mean temperature profile on which random fluctuations were superimposed. To reduce the computational costs, the long transient from fluid at rest to fully developed flow was run on a coarser grid. The final 3D results were interpolated to the finer mesh and the simulation continued to gain results for a certain time interval in which the flow is statistically stationary and which can be used for statistical analysis. Nevertheless, the total CPU-time on the Fujitsu VP400 at the Nuclear Research Centre Karlsruhe for this run was about 60 hours. The simulation for sodium, $Ra = 24{,}000$, was started from the final data of a previous simulation ($Ra = 12{,}000$, $Pr = 0.006$) on the same grid. For the simulation of air final results of a former air simulation [22] at $Ra = 380{,}000$ were interpolated and used as initial data.

SIMULATION RESULTS

VERIFICATION. The simulation with air is included here to gain a more detailed verification of the new semi-implicit time-integration scheme. The numerical results are compared to those of an experiment for the same Rayleigh number [23]. Vertical profiles of mean temperature and rms-values of the vertical velocity are given in Fig. 1. For reasons of symmetry only the lower half of the channel ($x_3 = 0$: lower wall, $x_3 = 1$: upper wall) is shown. The numerical results agree well with the experimental ones except for u_{3rms} near the wall. This discrepancy may be a consequence of inaccurate measurements because an unsystematic correlation coefficient between temperature and u_3 was found near the walls in the experiment.

Table 2. Verification of sodium simulations

Ra	Nusselt-number TURBIT	Nusselt-number experiment [24]
6,000	1.05 ± 0.015	1.10
24,000	1.37 ± 0.05	1.49

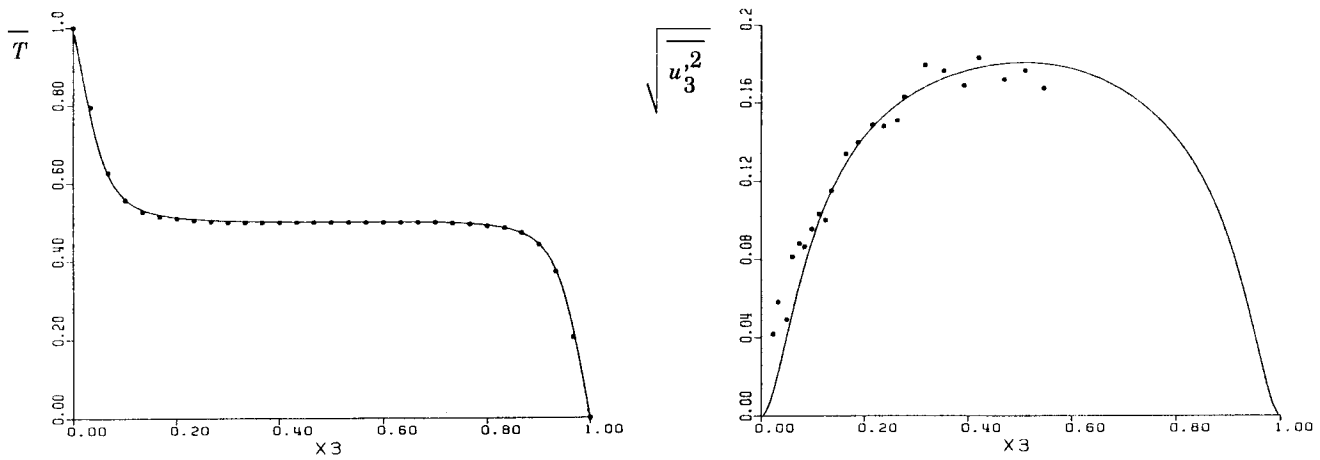


Fig. 1: Mean temperature profile and rms-value of vertical velocity in air: —TURBIT, ● experiment [23].

There are only very few experimental data available for Rayleigh-Bénard convection in liquid sodium which can be used for verification. The calculated Nusselt numbers compare roughly with the experiments of Kek [24], see Table 2. The small values of Nu show that most heat is transferred by conduction, not by convection. Better agreement is achieved for the rms-value of temperature fluctuations in channel midwidth. The results given in Fig. 2 in terms of the wall temperature difference agree well with the band of experimental data given by Kek.

FLOW STRUCTURES. As mentioned before, we don't have any detailed information and experimental evidence on the flow structures to be expected and on the corresponding velocity fields. We only know from stability analysis that the flow shall be time-dependent [25] and might be fully turbulent. From two-dimensional theoretical work [26] it is postulated that a solid body rotation of vortices shall occur at $Ra > 10^4$. From experiments indications were found that this kind of rotation really may exist, but it is found at smaller Rayleigh numbers [27].

Our recent direct simulations for Rayleigh-Bénard convection in sodium provide the first physically realistic information about the flow structures, mechanisms, and macroscopic wavelengths for this flow. Fig. 3 is an example for the static analysis of Rayleigh-Bénard convection in sodium at a Rayleigh number of 6,000. The isosurfaces indicate the locations at which the correlation $(T-0.5) * u_3$ is near zero, with T = local temperature between 0 and 1, u_3 = vertical velocity. This correlation tends to zero near midplane where T = 0.5; here one finds long band-like structures in the temperature field indicating the existence of regular rolls as in laminar convection. Additional analysis [28] confirms that the solid body rotation exists, but it is found only locally and for short times, and it vanishes more and more with increasing Rayleigh number because the large-scale vortex system, which is superimposed to the small scale turbulence, shows increasing three-dimensional distortions. The correlation plotted in Fig. 3 also tends to zero near the lower and upper walls where u_3 tends to zero; there the bands show rough boundaries and highly irregular spoke patterns across the rolls indicating that the velocity field is highly turbulent and has very small scales. The latter result is also shown by different colours which represent the local values of u_3. Thus this single picture represents both main features of this flow: The temperature field is largely controlled by molecular conduction, whereas the velocity field is highly turbulent. Indeed, all our simulations for sodium show three-dimensional aperiodic results obeying all statistical features of a highly turbulent velocity field. This means, the turbulence models for the velocity field have to account for the high turbulence level, whereas the models for the thermal energy field have mainly to account for molecular conduction and for convection at large scales.

Fig. 3: Isosurface for $(T-0.5)*u_3 = 0.006$ in Rayleigh-Bénard convection, colour code for vertical veloctiy, Ra = 6,000, Pr = 0.006, 120·120·19 mesh cells.

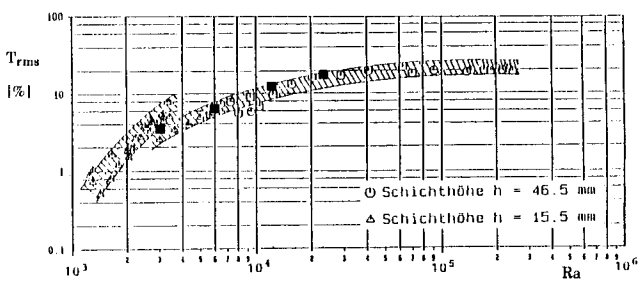


Fig. 2: RMS-values of temperature fluctuations in liquid sodium in the centre of the layer as function of the Rayleigh number.
(O, Δ = results of Kek [24], ■ results of TURBIT)

BALANCE EQUATION OF TURBULENT HEAT FLUX. The turbulent heat flux $\overline{u_3'T'}$ in the vertical direction is modelled in standard k-ε models [29] on the basis of an eddy diffusivity of heat a_t. Usually the concept of turbulent Prandtl number $Pr_t = \nu_t/a_t$ is used to calculate a_t from the eddy diffusivity of mass and from the empirically established coefficient Pr_t. This concept gives only poor results for natural convection and especially for liquid metals because it simply transfers features of turbulence from the velocity to the temperature field [7]. For liquid metals, however, turbulence features of the temperature and velocity fields are clearly different.

More promising concepts than that of using a turbulent Prandtl number are based on modelled forms of the transport equation for the turbulent heat flux. For Rayleigh-Bénard convection the transport equation for $\overline{u_3''T''}$ reduces to

$$0 = -\frac{\partial}{\partial x_3}\Bigg(\underbrace{\overline{u_3'^2 T'} + \overline{p'T'}}_{D_t} \underbrace{- \frac{1}{Pr\sqrt{Gr}}\,\overline{u_3' \frac{\partial T'}{\partial x_3}} - \frac{1}{\sqrt{Gr}}\,\overline{T' \frac{\partial u_3'}{\partial x_3}}}_{D_m} \Bigg)$$

$$\underbrace{- \overline{u_3'^2}\,\frac{\partial \overline{T}}{\partial x_3}}_{P} + \underbrace{\overline{T'^2}}_{} + \underbrace{\overline{p' \frac{\partial T'}{\partial x_3}}}_{PS} \qquad (2)$$

$$\underbrace{- \frac{1}{\sqrt{Gr}}\left(1 + \frac{1}{Pr}\right) \overline{\frac{\partial u_3'}{\partial x_i} \cdot \frac{\partial T'}{\partial x_i}}}_{MD} .$$

Here D denotes the diffusion of $\overline{u_3''T''}$, consisting of a turbulent part (D_t) and a molecular part (D_m). Responsible for the production of $\overline{u_3''T''}$ are both the mean temperature field (first part of P) and the buoyancy force (second part of P). The molecular destruction MD vanishes only for isotropic turbulence. In this case the pressure scrambling term PS is the only dissipation term [30].

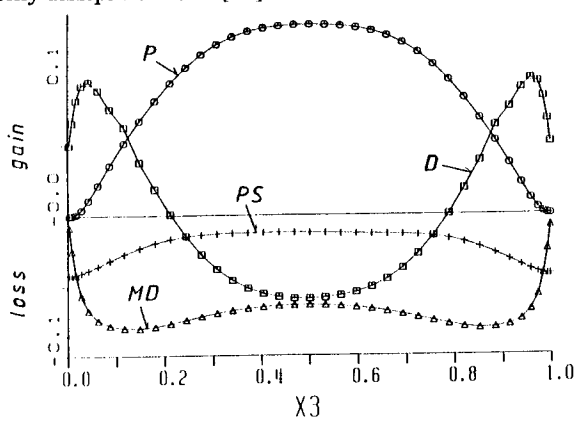


Fig. 4: Terms of the $\overline{u_3''T''}$ - equation in liquid sodium (Ra = 24,000):
□ diffusion, o production, Δ molecular destruction, + pressure scrambling term.

A direct comparison of the balance of turbulent heat flux between air and sodium simulations is not meaningful, since the vertical profiles of $\overline{u_3''T''}$ are different for both fluids. In the following we concentrate on sodium case Ra = 24,000. The vertical profiles of the different terms, given in Fig. 4, show, that not the pressure scrambling term but the molecular destruction is the main sink of turbulent heat flux. In the corresponding balance of the simulation with air [31] the molecular destruction is of minor importance, as compared to the pressure scrambling term. However, it is not zero because the flow is not isotropic. For liquid metals thermal diffusion dominates the temperature field, therefore the molecular destruction plays such an important role for this type of fluid.

The production term P is zero at the walls and increases towards the centre of the channel. Here we find a surplus of P over the molecular destruction and pressure scrambling term. This surplus is balanced by the diffusion D which redistributes turbulent heat flux from the centre of the channel towards the wall regions.

Terms that have to be modelled to close the transport equation for the turbulent heat flux are the diffusion and the pressure scrambling terms. Here we consider only closure assumptions for diffusive transport. Usually molecular diffusion and diffusion by pressure fluctuations are neglected [32]. Therefore only the correlation $\overline{u_3'^2T'}$ has to be modelled. In Fig. 5 we compare vertical profiles of $\overline{u_3'^2T'}$ and $\overline{p'T'}$ and find, that in Rayleigh-Bénard convection turbulent diffusion is mainly caused by pressure fluctuations. Thus $\overline{p'T'}$ seems to be an important term that must be modelled adequately.

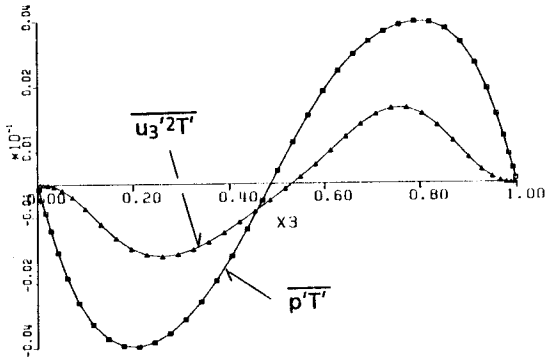


Fig. 5: Turbulent diffusion terms in liquid sodium (Ra = 24,000): □ $\overline{p'T'}$, Δ $\overline{u_3'^2T'}$.

The model commonly used for closure of turbulent diffusion is [30]

$$- \overline{u_3'^2 T'} = 2\, c_T \frac{k}{\varepsilon}\, \overline{u_3'^2}\, \overline{\frac{\partial u_3' T'}{\partial x_3}}, \qquad (3)$$

where C_T is approximately 0.1. In Fig. 6 we show the vertical profile of C_T evaluated from our numerical results. Additionally the vertical profile of

$$C_{TS} = \frac{- \left(\overline{u_3'^2 T'} + \overline{p'T'} \right)}{2 \frac{k}{\varepsilon}\, \overline{u_3'^2}\, \overline{\frac{\partial u_3' T'}{\partial x_3}}} \qquad (4)$$

is given, which implies that both, turbulent diffusion by velocity and by pressure fluctuations, are modelled in common by the right hand side of Equation (3). C_T and C_{TS} are almost constant through the channel. Only near the walls a strong increase is observed. The small peaks found at channel midwidth can be explained by the vanishing gradient of turbulent heat flux, appearing in the denominator. In general, the model assumption given above seems to be a reasonable approximation for turbulent diffusion, but additional wall corrections should be used. However, the values $C_T \approx 0.008$ and $C_{TS} \approx 0.02$ resulting from the analysis are about a magnitude lower than the ones commonly used. Furthermore, the gradient diffusion concept may in principle be inadequate because it implies that in case of constant turbulent heat flux there can be no diffusion. Results for the simulation with air reveal, however, that in the centre of the channel there is a diffusive transport, though the turbulent heat flux is constant in this region [31].

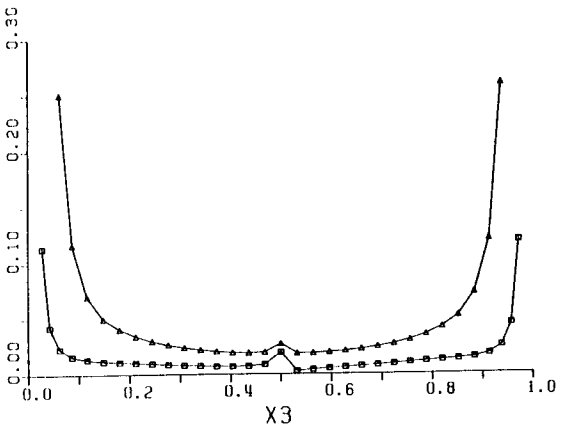


Fig. 6: Coefficients of turbulent diffusion model of $\overline{u_3'T'}$: □ C_T, Δ C_{TS}.

CONCLUSIONS

The results of direct simulations of turbulence discussed above demonstrate the advantages and limitations of the method to provide detailed insight into mechanisms of turbulence and its statistics. The method can be used to investigate any statistical turbulence data at small turbulence levels. Here, all terms of the turbulent heat flux equation were analysed for Rayleigh-Bénard convection. It turned out that diffusion by pressure fluctuations dominates over that by velocity fluctuations. Therefore existing statistical turbulence models need additional assumptions to account for the diffusion by pressure fluctuations. Capabilities of current computers are not sufficiently advanced to simulate high turbulence levels with the direct method which are of interest for practical applications in engineering. These problems can be better treated by the large eddy simulation method. It is not as universal as the direct method but more advanced than the statistical one. The short overview on reactor related applications of both methods indicates that at least the large eddy simulation method was used for some prototypical engineering problems. It is just going to be used for analysing current engineering problems in nuclear reactor thermal-hydraulics. It might become the appropriate method to analyse time-dependent problems caused by turbulence like thermal striping.

ACKNOWLEDGEMENTS

The tools for statistical analysis of the transport equation of turbulent heat flux were provided by I. Hiltner for grafical analysis of the flow mechanisms by E. Hesselschwerdt. Their work is gratefully acknowledged. The author G. Grötzbach gratefully acknowledges the financial support by FAPERGS (Fundacao de Amparo a Pesquisa do Rio Grande do Sul), by UFRGS (Universidade Federal do Rio Grande do Sul), and by the DAAD (Deutscher Akademischer Austauschdienst, Bonn) enabling this presentation at ENFIR.

REFERENCES

[1] SATOH, K.; MIYAKOSHI, H. Study of decay heat removal by natural circulation. Proc. NURETH-4, (1989), pp. 378-383.

[2] HOFFMANN, H.; KAMIDE, H.; MARTEN, K.; OHSHIMA, H.; WEINBERG, D. Investigations on the transition from forced to natural convection for the pool type EFR in the 3d RAMONA model. Int. Conf. on Fast Reactors and Related Fuel Cycles, Oct. 28 - Nov. 1, 1991, Kyoto.

[3] NINOKATA, H. Advances in computer simulation of fast breeder reactor thermalhydraulics. Proc. SNA '90, (1990), pp.80-85.

[4] BORGWALDT, H.A. CRESOR, a robust vectorized Poisson solver implemented in the COMMIX-2(V) thermal-hydraulics code. Proc. SNA '90, (1990), pp. 346-351.

[5] LAWRENCE, S.P. Turbulence modelling in naturally convecting fluids. Culham Lab., CLM-R292, Sept. 1989.

[6] GRÖTZBACH, G. Direct numerical and large eddy simulation of turbulent channel flows. Encyclopaedia of Fluid Mechanics, Gulf Publ. Houston, Vol. 6, (1987), pp. 1337-1391.

[7] GRÖTZBACH, G.; WÖRNER, M. Analysis of second order transport equations by numerical simulations of turbulent convection in liquid metals. Proc. NURETH-5, (1992), pp. 358-365.

[8] WÖRNER, M.; GRÖTZBACH, G. Analysis of semi-implicit time integration schemes for direct numerical simulation of turbulent convection in liquid metals. In: Notes on Numerical Fluid Mechanics, Vol. 35, Ed. J.B. Vos, A. Rizzi, I.L. Ryhming, Vieweg Verlag Braunschweig (1992), pp. 542-551.

[9] WÖRNER, M. Direkte Simulation turbulenter Rayleigh-Bénard Konvektion in flüssigem Natrium. Dissertation, Universität Karlsruhe, KfK 5228, Kernforschungszentrum Karlsruhe, 1993.

[10] SCHMIDT, H.; SCHUMANN, U.; VOLKERT, H.; ULRICH, W. Three-dimensional, direct and vectorized elliptic solvers for various boundary conditions. DFVLR-Mitteilung 84-15, (1984).

[11] SCHUMANN, U. Numerical investigation of the wall pressure fluctuations in channel flow. Nucl. Engng. Design 32, pp. 37-46 (1975)

[12] PRUITT, J.M.; HASSAN, Y.A.; STEININGER, D.A. Prediction of steam generator turbulence using large-eddy simulation. Transaction of the ANS, Vol. 66, 1992, pp. 563-564.

[13] GRÖTZBACH, G. Numerical investigation of radial mixing capabilities in strongly buoyancy-influenced vertical, turbulent channel flows. Nucl. Engng. Design, 54, pp. 49-66 (1979)

[14] GAO, S.; LESLIE, D.C.; VOKE, P.R. Large eddy simulation of thermal impinging jet. University of Surrey, Dept. of MechanicalEngng, ME-FD/91.02, 1991.

[15] LAWN, C.J. Turbulent temperature fluctuations in liquid metals. Int. J. Heat Mass Transfer, 20, pp. 1035-1044 (1977)

[16] GRÖTZBACH, G. Numerical simulation of turbulent temperature fluctuations in liquid metals. Int. J. Heat Mass Transfer, 24, pp. 475-490 (1981)

[17] GRÖTZBACH, G. Numerical Methods in Laminar & Turbulent Flow. Ed. C. Taylor, B.A. Schrefler, Pineridge, pp. 593-604 (1981)

[18] GRÖTZBACH, G. Direct numerical simulation of the turbulent momentum and heat transfer in an internally heated luid layer. Heat Transfer 1982, Eds. U. Grigull et.al. 2, Hemisphere, pp. 141-146 (1982)

[19] GRÖTZBACH, G. Turbulent heat transfer in an internally heated fluid layer. Refined Flow Modelling and Turbulence Measurements. Eds. Y. Iwasa et al., Un. Ac. Press, pp. 267-275 (1989)

[20] KIKUCHI, Y.; SHIOYAMA, T.; KAWARA, Z. Turbulent heat transport in a horizontal fluid layer heated internally and from below. Int. J. Heat Mass Transfer 29, pp. 451-461 (1986)

[21] GRÖTZBACH, G. Spatial resolution requirements for direct numerical simulation of the Rayleigh-Bénard convection. J. Comp. Phys. 49, (1983), pp. 241-264.

[22] GRÖTZBACH, G. Simulation of turbulent flow and heat transfer for selected problems of nuclear thermal-hydraulics. The First Int. Conf. on Supercomputing in Nuclear Applications; Ed. Japan Atomic Energy Research Institute, Tokyo; Publ. Nuclear Energy Data Center, Tokai-mura, Japan, 1990, pp. 29-35.

[23] DEARDORFF, J.W.; WILLIS, G.E. Investigation of turbulent thermal convection between horizontal planes. J. Fluid Mech., Vol. 28, (1967), pp. 576-704.

[24] KEK, V. Bénard Konvektion in flüssigen Natrium-schichten. Dissertation, Universität Karlsruhe, KfK 4611, Kernforschungszentrum Karlsruhe, 1989.

[25] CLEVER, R.M.; BUSSE, F.H. Convection at very low Prandtl numbers. Physics of Fluids, A2, 1990, pp. 334-339.

[26] CLEVER, R.M.; BUSSE, F.H. Low Prandtl number convection in a layer heated from below, J. Fluid Mech. 102, 1981, pp. 61-74.

[27] CHIFFAUDEL, A.; FAUVE, S.; PERRIN, B. Viscous and inertial convection at low Prandtl number: Experimental study. Europhysics Letters 4, 1987, pp. 555-560.

[28] GRÖTZBACH, G.; WÖRNER, M. Analysis of flow mechanisms in Rayleigh-Bénard convection at small Prandtl numbers. Joint Int. Conf. on Mathematical Methods and Supercomputing in Nuclear Applications, April 19-23, 1993, Karlsruhe, Germany, Proc. Ed. H. Küsters, E. Stein, W. Werner, Mathematical Methods and Supercomputing in Nuclear Applications, Kernforschungszentrum Karlsruhe, 1993, Vol. 1, pp. 236-247.

[29] RODI, W. Turbulence models and their application in hydraulics - a state of the art review. IAHR-publication, Delft 1980.

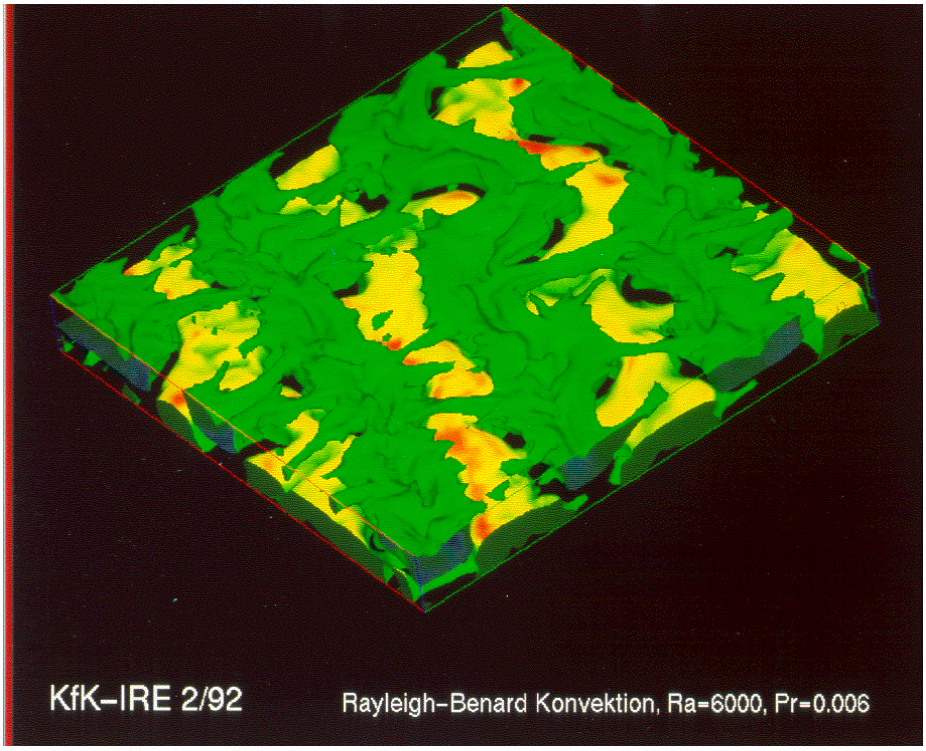
[30] HOSSAIN, M.S.; RODI, W. A turbulence model for buoyant flows and its application to vertical buoyant jets. In Turbulent jets and plumes; Ed. W. Rodi, Pergamon Press (1982).

[31] WÖRNER, M.; GRÖTZBACH, G. Turbulent heat flux balance for natural convection in air and sodium analysed by direct numerical simulations. Fifth Int. Symposium on Refined Flow Modelling and Turbulence Measurements, Sept. 7-10, 1993, Paris.

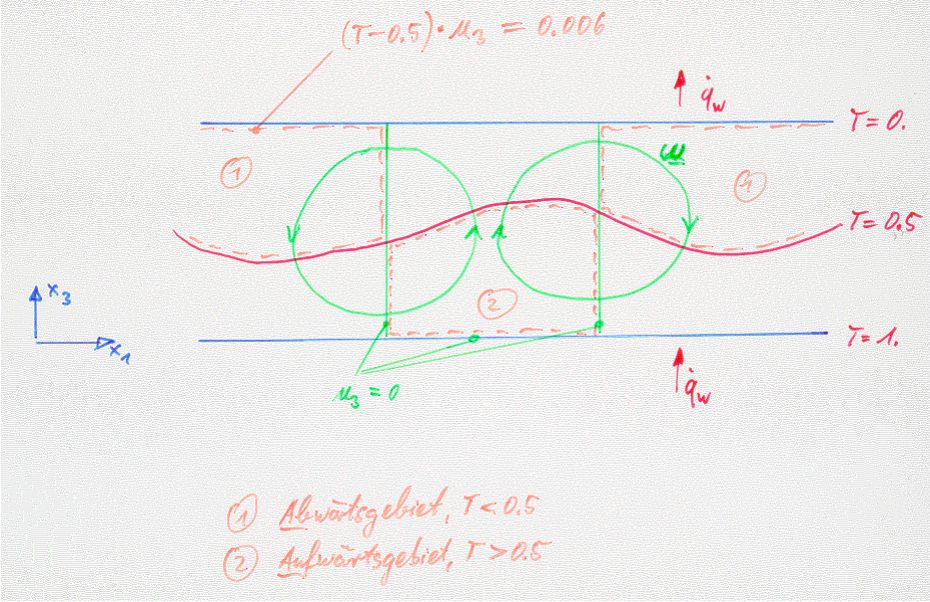
[32] LAUNDER, B.E. Heat and Mass Transport, in Topics in Applied Physics, Vol. 12, Ed. P. Bradshaw, Springer (1976).

Supplement for Figure 3
in colour and with explenation

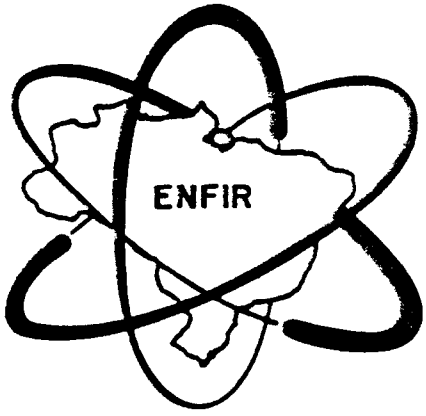
Isosurface for $(T - 0.5)*u_3 = 0.006$
with colour code for vertical velocity (red upward, blue downward)



The isosurface is formed by the dashed line,
i.e. at T=0.5
and at $u_3 = 0.006$ (vertical component).



Thus, near the lower wall the upward flow is inside the isosurface,
and near the upper wall the downward flow is inside.



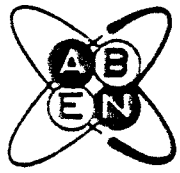
IX ENCONTRO NACIONAL DE FÍSICA DE REATORES E TERMO-HIDRÁULICA

9th BRAZILIAN MEETING ON REACTOR PHYSICS AND THERMAL HYDRAULICS

25-29 Outubro 1993
Hotel Glória
Caxambu, MG - Brasil

ANAIS
PROCEEDINGS

CNEN
Centro de Desenvolvimento da Tecnologia Nuclear



ABEN - Associação Brasileira de Energia Nuclear