
**HEAT AND MASS TRANSFER
AND PHYSICAL GASDYNAMICS**

Numerical Simulation of Turbulent Convection of Air in a Square Cavity Heated on the Side

A. G. Abramov and E. M. Smirnov

St. Petersburg State Polytechnical University, St. Petersburg, 195251 Russia

Received December 17, 2004

Abstract—A set of three-dimensional unsteady-state equations is used, with the Rayleigh number of 1.58×10^9 , to calculate the turbulent, statistically two-dimensional convection of air in a closed cavity with side walls heated to different temperatures. The turbulence is simulated using the equation of transfer of kinetic energy of turbulent motion, with the dissipation rate calculated differently in the vicinity of and away from the walls. The calculation results are compared with available data of experimental and numerical investigations of other authors. The possibilities are discussed of using this approach to predict averaged characteristics of free convection.

INTRODUCTION

The development of methods of calculation of turbulent natural convection in cavities heated on the side still remains an urgent problem from the standpoint of both basic research and practical applications. Until recently, numerical investigations of this problem involved the use of averaged Navier–Stokes or Reynolds equations (Reynolds-Averaged-Navier-Stokes, RANS). In so doing, the calculation results depended significantly on the choice of the closing model of turbulence, and acceptable agreement with experimental data for a number of integral characteristics could only be obtained by special adjustment of the models to fit the class of problems being treated.

At present, the method of direct numerical simulation (DNS) [1, 2] and the method of large eddy simulation (LES) [3] are used in solving three-dimensional unsteady-state problems in turbulent convection. The present-day capabilities of the DNS method with the determination of all components of motion are limited to calculations at relatively low values of the Rayleigh number of the order of 10^8 or lower. Compared to DNS, the LES method may be used to calculate flows with much higher values of the Rayleigh number. However, the use of the LES method for the calculation of wall flows requires either the introduction of additional “empiricism”, which lies in parametrization of the layer adjoining the wall, or the use of com-

putational grids whose characteristics approach those of the grids of the DNS method [4, 5].

Nowadays, combined (hybrid) approaches to the simulation of turbulence are growing in popularity; these approaches combine the advantages of the LES and RANS methods. The most significant results in this respect were obtained using the hybrid approach referred to as the method of “detached” eddy simulation (DES) [5]. The central idea of this approach is that the wall layers are calculated by the unsteady-state RANS method, and the regions away from the walls – by the LES method. In so doing, the LES method involves the use of the same model of turbulence for the calculation of the turbulent characteristics as that used in the RANS region, but with an over-determined characteristic scale of length, which provides for the desired level of dissipation of turbulent energy. A fairly detailed description of the results of using the “standard” DNS method to calculate flows with extensive regions of flow separation is found in [6, 7]. As a rule, within the framework of DES, the boundary layer or its major part is calculated using the RANS method, which is largely attained owing to the structure of the computational grid. On the contrary, in the case of the “nonstandard” DES method used in the calculations of internal flows [8, 9], only a part of the wall shear layer is calculated by the RANS method. This approach may be regarded as the LES method involving a special technique of resolving the regions

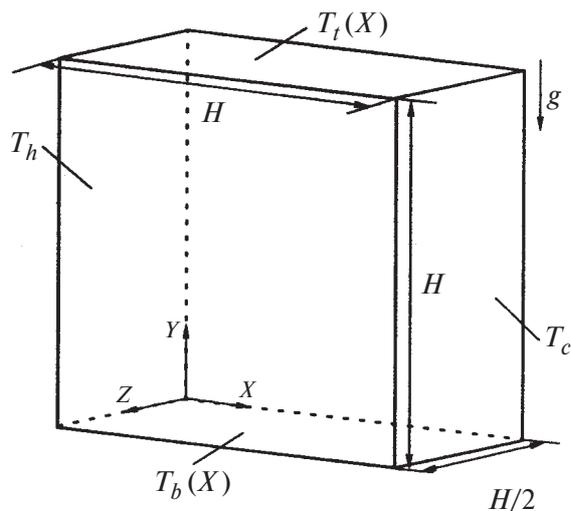


Fig. 1. Schematic of the cavity and the boundary conditions.

in the vicinity of the walls, which is an alternative to the method of wall functions.

The results of using the nonstandard DES method based on the use of a single equation for the kinetic energy of turbulence to calculate the Rayleigh–Bernard turbulent convection in closed cavities of simple shape are given in recent publications [10, 11]. We investigate the possibilities of using a hybrid approach similar to that of [10, 11] to perform numerical simulation of the characteristics of turbulent convection of air in a closed square cavity with side walls heated to different temperatures.

FORMULATION OF THE PROBLEM AND BASIC EQUATIONS

The calculations, whose results are given below, were performed for the conditions of a recent experimental investigation [12]. We treated a statistically two-dimensional natural convection of air in a square cavity with (opposite) vertical walls whose temperatures T_h and T_c are maintained constant ($T_h > T_c$) and with heat-conducting horizontal walls whose temperatures $T_b(x)$ and $T_t(x)$ correspond to experimentally measured distributions [12] (Fig. 1). Note that a cavity of dimensions $H \times H \times H/2$ with non-heat-conducting vertical end walls was used in the experiments. Tian and Karayiannis [12] demonstrated that the convection in the middle part of the cavity may be treated as statistically two-dimensional and uniform in the direction of the end walls.

In view of this, we assume that the time average flow occurs in a plane (X, Y) , with the vertical axis Y directed in opposition to the vector of gravitational acceleration. In view of the inferences made by Peng and Davidson [3], who applied the LES method for solving the same problem, we take the extent of the computational domain in the uniform direction Z to be equal to half the height of the cavity. The conditions of periodicity were preassigned on the boundary planes $Z = 0$ and $Z = H/2$.

Within the Boussinesq approximation, we represent the set of dimensionless equations, which describe unsteady-state fields of velocity, pressure, and temperature, in the form

$$\nabla \cdot \mathbf{V} = 0,$$

$$\frac{\partial \mathbf{V}}{\partial t} + (\nabla \cdot \mathbf{V})\mathbf{V} = -\nabla p^* + 2\nabla(\nu_{\text{eff}}\dot{\mathbf{S}}) - T\mathbf{e}_g, \quad (1)$$

$$\frac{\partial T}{\partial t} + (\nabla \cdot \mathbf{V})T = \nabla(a_{\text{eff}}\nabla T).$$

Here, the scales of length, velocity, and temperature are provided by the cavity height H , the rate of buoyancy V_b , and the temperature difference $\Delta T = T_h - T_c$, respectively. The time is normalized to the ratio H/V_b . The eddy (turbulent) viscosity ν_t determined by one or another model of turbulence appears in the expression for effective dimensionless coefficient of viscosity $\nu_{\text{eff}} = \sqrt{\text{Pr}/\text{Ra}} + \nu_t$. The effective coefficient of thermal diffusivity in Eq. (1) is written as $a_{\text{eff}} = 1/\sqrt{\text{Pr}\text{Ra}} + \nu_t/\text{Pr}_t$, where $\text{Pr}_t = 0.4$ is the turbulent Prandtl number. As in the experiment of Tian and Karayiannis [12], the Rayleigh number Ra was taken to be 1.58×10^9 , and the Prandtl number $\text{Pr} = 0.71$.

SIMULATION OF TURBULENCE

The hybrid approach to simulation of turbulence employed by us is based on a differential model with one equation describing the transfer of kinetic energy k of turbulent motion [9–11]. As in the case of the standard DES method, the model provides for the solution of unified equation for k in the entire computational domain. In the LES model, the equation for kinetic energy of subgrid motion is solved, and in the

RANS model – the equation for total kinetic energy of turbulence. The equation for k is written as

$$\frac{dk}{dt} = \frac{\partial}{\partial x_j} \left[(v + v_t) \frac{\partial k}{\partial x_j} \right] + P_k - \bar{\varepsilon},$$

where $P_k = 2v_t \bar{S}_{ij} \bar{S}_{ij}$ is the generation term, and $\bar{\varepsilon} = \max\{\varepsilon^{\text{RANS}}, \varepsilon^{\text{LES}}\}$ is the dissipation term. The “switching” between the RANS and LES domains is made on the principle of the choice of maximum from two values of $\bar{\varepsilon}$ calculated by two methods,

$$\begin{aligned} \varepsilon^{\text{RANS}} &= k^{3/2}/(lF_\varepsilon), \quad l = \kappa C_\mu^{-3/4} y, \\ F_\varepsilon &= f(\text{Re}_y), \quad \text{Re}_y = \sqrt{ky}/\nu; \\ \varepsilon^{\text{LES}} &= C_\varepsilon k^{3/2}/\bar{\Delta}, \quad \bar{\Delta} = \sqrt[3]{\Delta_1 \Delta_2 \Delta_3}. \end{aligned} \quad (2)$$

According to relation (3), the characteristic (undamped) linear scale of turbulence for the RANS domain is taken to be proportional to the distance to the nearest wall. The turbulent viscosity is calculated as

$$\nu_t = C_\mu f_\mu k^2 / \bar{\varepsilon}.$$

The functions F_μ and f_μ are preassigned as in the semiempirical model of turbulence of Wolfshtein [13],

$$\begin{aligned} F_\varepsilon &= 1 - \exp(-\text{Re}_y/A_\varepsilon), \\ f_\mu &= 1 - \exp(-\text{Re}_y/A_\mu). \end{aligned} \quad (3)$$

In order to provide for continuous variation of turbulent viscosity in the vicinity of the interface between the LES and RANS domains, the damping function f_μ in the LES model is written in a form other than (5), namely,

$$\begin{aligned} f_\mu &= 1 - \exp[-\text{Re}_t/A_\mu (\kappa C_\mu^{-3/4} F_\varepsilon(\text{Re}_y))^{-1}], \\ l_t &= \min\{lF_\varepsilon, \bar{\Delta}/C_\varepsilon\}, \quad \text{Re}_t = \sqrt{k}l_t/\nu. \end{aligned}$$

The empirical constants of the model were taken to be as follows: $C_\mu = 0.09$, $C_\varepsilon = 0.75$, $\kappa = 0.41$, $A_\mu = 10$, and $A_\varepsilon = 5.1$.

COMPUTATIONAL ASPECTS

The calculations involved the use of the SINF (Supersonic to Incompressible Flows) computer codes developed during a period of over ten years at the Chair of Aerohydrodynamics of the St. Petersburg Polytechnical University and designed for solving three-dimensional Navier–Stokes equations. This

software package makes it possible to perform calculations of steady-state, unsteady-state, subsonic, and supersonic flows of liquid or gas developing in regions of complex geometry [14]. The numerical method is based on the use of multiblock structured nonuniform grids matching the boundaries of the region of flow. The spatial quantization is performed by the method of finite volumes with the second order of accuracy. The artificial compressibility method is used to calculate the flows of incompressible liquid and low-velocity flows of gas. The advance with respect to relaxation time is accomplished using implicit schemes of the type of the approximate factorization method or Gauss–Seidel relaxation schemes. An implicit three-level scheme of the second order of accuracy is used to solve unsteady-state problems. In this case, the iteration process with respect to relaxation time is organized on each level with respect to physical time. The SINF computer codes include a number of high- and low-Reynolds versions of one- and two-parameter models of turbulence, which enables one to perform trade-off studies of turbulent flows using the RANS method. In 2000–2002, the basic version of SINF was supplemented with the possibility of describing the turbulence with the aid of the LES and DES methods. The parallel version of SINF, which is used in calculations with high-efficiency cluster systems, provides for the division of the computational domain into subdomains (blocks) and for the use of the SPMD (Single Program Multiple Data) strategy and MPI (Message Passing Interface) standard [15].

The calculations of convection in a square cavity involved the use of a nonuniform grid containing 157 thousand cells, with the nodes crowding towards all walls of the cavity so that the dimensionless coordinate Y^+ of the centers of grid cells closest to the wall was less than unity.

The dimensionless time step was taken to be 0.05. Zero values of relative velocity and average value of dimensionless temperature were preassigned as initial. The solution was obtained for an interval of 200 time units. In determining time average characteristics, the initial period of development of convection with a duration of 50 time units was discarded.

Within the employed method of description of turbulent convection, the set of RANS equations was solved from the wall to values of $Y^+ \approx 15$ ($Y \approx 0.01$)

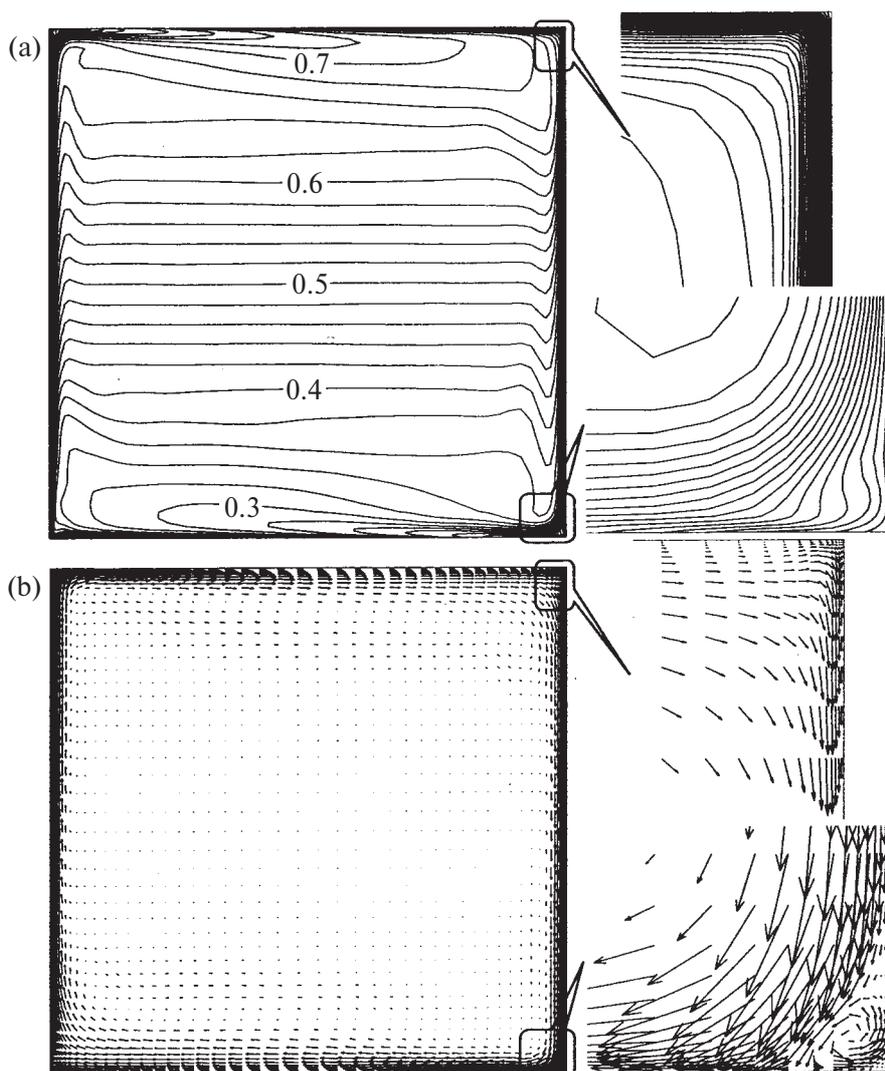


Fig. 2. Averaged fields of (a) velocity and (b) temperature in the vertical cross section of the cavity and fragments of flow in the corners of the cavity.

in nine or ten nodes of computational grid. It was here that the maximal values of turbulent viscosity were observed, which were three or four times the values of molecular viscosity. Note that the computational grid cells in the wall region had a highly extended shape, so that the size of cells closest to the wall in the vicinity of the central cross sections of the cavity ($X = 0.5$ and $Y = 0.5$) reached the value of one hundred. In the case of such grids, the models of subgrid stresses in the LES method cannot provide for an adequately exact resolution of eddy structures in the vicinity of the walls, which are dynamically important to the flow. In the hybrid approach employed by us, the LES method is used away from the walls, where the spatial anisotropy of grid cells is not so pronounced.

CALCULATION RESULTS AND DISCUSSION

Figure 2 illustrates the calculated (time and direction Z average) fields of velocity and temperature in the plane (X, Y) . One can see that the buoyancy-initiated flow is largely localized in relatively thin wall layers; in so doing, the upward flow in the vicinity of the hot wall “closes” with the downward flow in the vicinity of the cold wall by means of flows along the horizontal walls. The drawing-in effect of intense jets forming at the horizontal walls causes the formation of recirculation structures extended along the walls. Eddies of relatively low intensity are formed in the corner made by the crossing of the vertical cold wall and lower horizontal wall and in the opposite (diago-

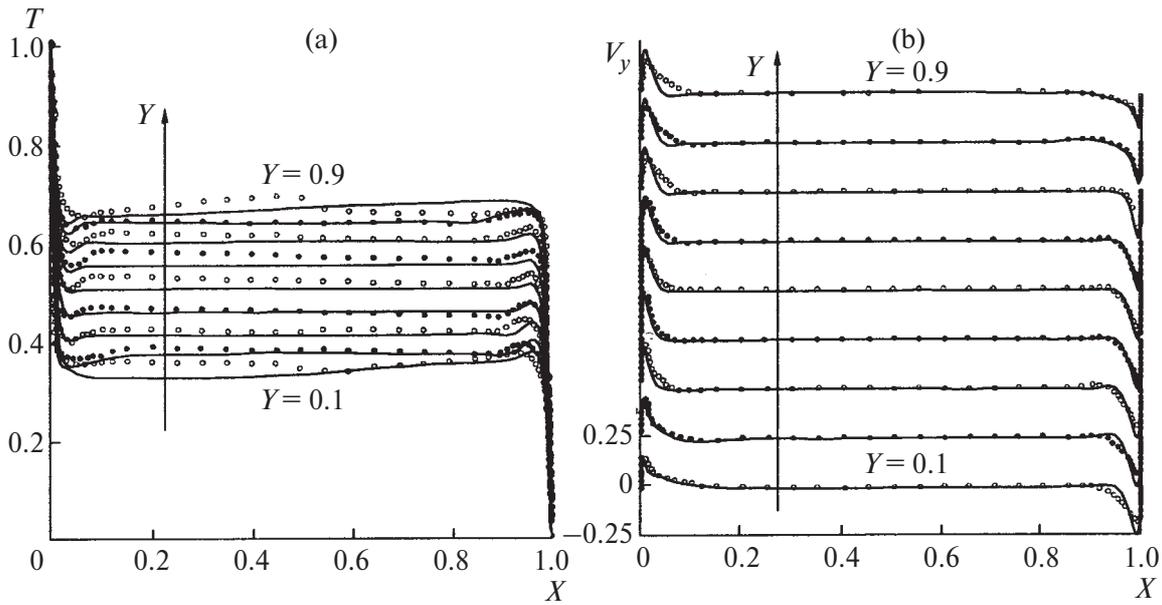


Fig. 3. Profiles of (a) temperature and (b) vertical component of velocity at different distances Y from the cavity bottom. Curves – our calculation, points – experiment [12].

nally) corner. The maximal temperature gradients are observed at the vertical walls of the cavity. The isotherms in the central part of the cavity are almost horizontal and distributed uniformly over the cross section. The variation of temperature with height corresponds to stable stratification which suppresses the development of turbulence in the flow core. However, an unstable stratification of temperature is present in the boundary layers at the horizontal heat-conducting walls of the cavity. In so doing, the width of the unstably stratified region increases with the development of wall jets, where vertical discharges of liquid occur in the form of thermals characteristic of Rayleigh–Bernard convection [10, 16]. By and large, the fields of velocity and temperature exhibit a clearly pronounced central symmetry. The calculated fields of velocity and temperature agree quite adequately with the experimental results of Tian and Karayiannis [12].

In Fig. 3, the profiles of temperature and vertical component of velocity at different distances from the cavity bottom calculated by us are compared with those measured by Tian and Karayiannis [12]. One can see in Fig. 3 that, in both experiment and calculation, the thickness of thermal boundary layer increases monotonically with the development of flow at the isothermal walls. At the same time, the thickness of velocity boundary layer and the maximal value of the vertical component of velocity vary nonmonotonically: the increase in thickness in the direction of median horizontal cross section changes to decrease on

approaching the opposite upper horizontal wall. The experimentally obtained profiles of velocity and temperature agree well with the predicted ones in the middle part of the cavity; however, the calculation in the vicinity of horizontal walls gives an appreciably smaller thickness of boundary layers and maximal values of velocity overstated by 15–20%. Figure 4 shows, on an enlarged scale, the variation of velocity and temperature in the wall layer on a heated vertical wall. It follows from Fig. 4 that the calculated and measured profiles agree quite adequately in the vicinity of $Y = 0.5$. Note that the velocity boundary layer in

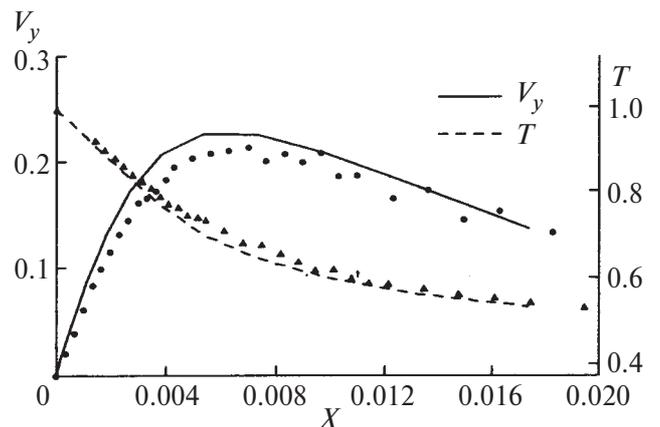


Fig. 4. Fragments of profiles of temperature and vertical component of velocity in the vicinity of the heated wall of the cavity in the cross section $Y = 0.5$. Curves – our calculation, points – experiment [12].

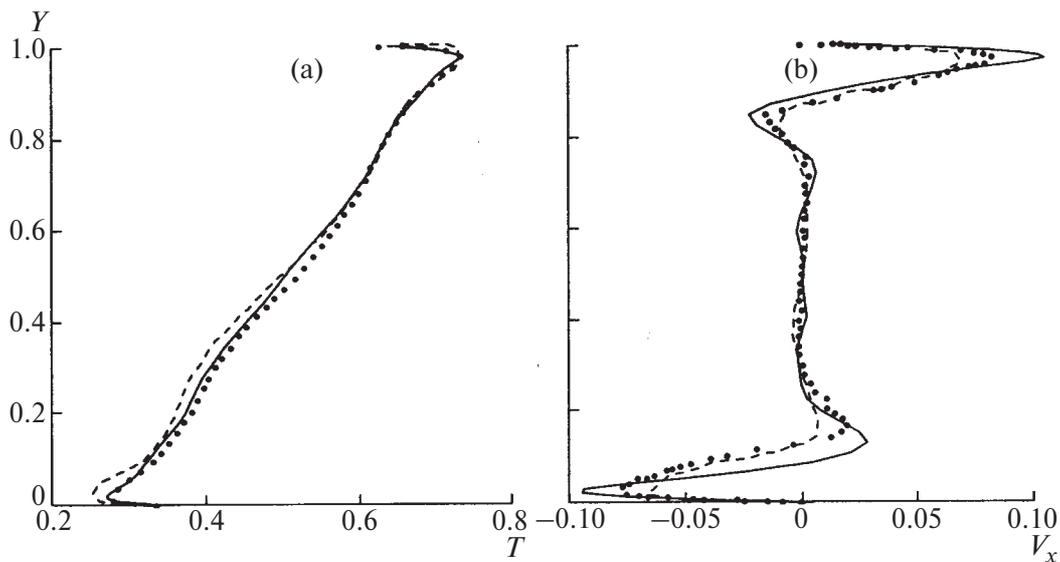


Fig. 5. The variation of (a) temperature and (b) vertical component of velocity V_x along a vertical line ($X = 0.5$). Continuous curves – our calculation, dashed curves – calculation by the LES method [3], points – experiment [12].

this region turns out to be much thinner than the temperature boundary layer. Note that the region of main growth of the vertical component of velocity, as well as a large part of the temperature boundary layer, found themselves in the zone in which the characteristics of turbulence are calculated by the RANS method.

The variation of temperature and horizontal component of velocity along a vertical line passing through the center of the cavity is shown in Fig. 5. As in the case of flow at vertical walls, the calculation produced somewhat overstated values of velocity $V_{x \max}$ in the vicinity of the horizontal boundaries. The maximal calculated values of $V_{x \max}$ are approximately twice lower than those of $V_{y \max}$, as is observed experimentally as well. The profiles given in Fig. 5 exhibit regions with the direction of velocity opposite to that of main flow; this is indicative of the presence of recirculation structures in the vicinity of the external boundary of jets forming at the horizontal walls. By and large, the results of our calculations are in as good an agreement with experiment as the data obtained using the “pure” LES method [3]. Note that the LES method employed in [3] involved the use of an algebraic dynamic model of subgrid turbulence on a grid containing 580 thousand cells.

The theoretically and experimentally obtained distributions of the friction coefficient C_f along the perimeter of the cavity are given in Fig. 6. The origin corresponds to $X = 0$ and $Y = 0$, i.e., to the bottom of

the hot wall. The maximal value of friction is attained in the sections $Y \approx 0.4$ and $Y \approx 0.6$ on the hot and cold vertical walls, respectively. The negative values of C_f correspond to the variation of the direction of liquid motion in the vicinity of two corners of the cavity where eddy structures are formed. One can see in Fig. 6 that the values of friction coefficient obtained by us and by Peng and Davidson [3] are somewhat overstated compared to the experimental data. However, the qualitative pattern of these distributions is the same.

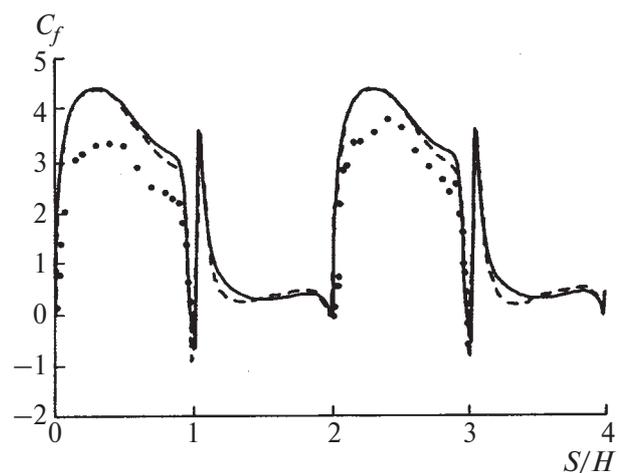


Fig. 6. The distribution of the friction coefficient along the perimeter of the cavity. Continuous curves – our calculation, dashed curves – calculation by the LES method [3], points – experiment [12].

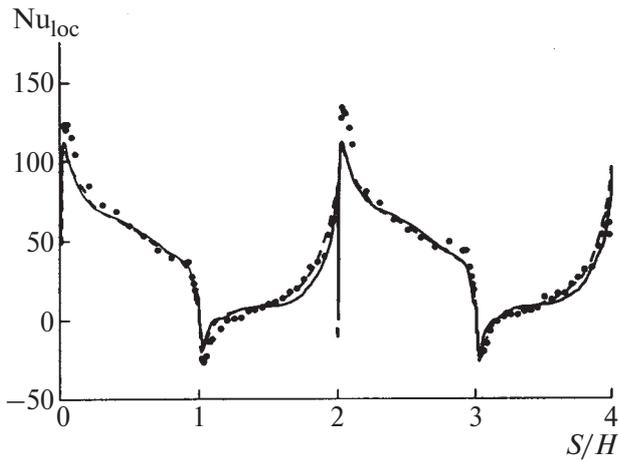


Fig. 7. The distribution of local dimensionless heat flux along the perimeter of the cavity. Continuous curves – our calculation, dashed curves – calculation by the LES method [3], points – experiment [12].

We will now consider the thermal characteristics of convection. Comparison of the theoretically and experimentally obtained distributions of the local Nusselt number Nu_{loc} (dimensionless heat flux) along the walls of the cavity is made in Fig. 7. One can see that the maximal values of local heat flux on the isothermal walls are observed at the bottom of the hot wall and at the top of the cold wall, where the thickness of temperature boundary layers is minimal. The air from the opposite (cold or hot, respectively) walls is delivered to these regions by horizontal wall jets. In accordance with the structure of temperature fields treated above, the local heat flux decreases as the main flow develops. In our calculations, as in the calculations by the LES method [3], the maximal values of Nu_{loc} are approximately 15% lower than the experimentally obtained values. The integral value of the Nusselt number Nu_v in our calculation turned out to be 60.1 compared to the experimentally obtained value of 64.6. The calculated distributions of the Nusselt number Nu_{loc} along the horizontal walls revealed the presence of experimentally observed regions with different directions of heat flux. The integral value of the Nusselt number Nu_h for the horizontal walls is 14.1 in calculation and 15.3 in experiment.

CONCLUSIONS

Analysis of the results of calculation of the turbulent, statistically two-dimensional convection of air in a closed cavity with side walls heated to different tem-

peratures with the Rayleigh number of 1.58×10^9 led to the following conclusions. The hybrid RANS/LES approach, which involves the use of the equation of transfer of kinetic turbulent energy, provided the possibility of employing relatively fine-mesh grids to reproduce all basic features of the fields of velocity and temperature, as well as of the distributions of the friction coefficient and local heat flux along the walls of the cavity. The predicted characteristics are in adequate agreement with the experimental data and with the results of calculations by the LES method obtained in a much finer computational grid. In view of the fact that the use of the hybrid RANS/LES approach for the simulation of the Rayleigh–Benard turbulent convection and for the calculation of the problem being treated produced results that are very close to experimental, this procedure may be suggested for use in numerical investigation of natural-convection flows in closed cavities under arbitrary thermal conditions on the boundaries.

ACKNOWLEDGMENTS

This study was supported by the Russian Program for support of leading scientific schools (grant NSh-1389.2003.8) and by the Russian Foundation for Basic Research (grant no. 05-02-17189-a).

NOTATION

a , thermal diffusivity; g , acceleration of gravity; e_g , unit vector in the direction of the effect of the force of gravity; H , height of the cavity; T_h and T_c , temperature of the hot and cold walls, respectively; $\Delta T = T_h - T_c$, scale of temperature variation; T , normalized temperature; $V_b = (g\beta\Delta TH)^{1/2}$, rate of buoyancy; \mathbf{V} , velocity vector; t , dimensionless time; p^* , reduced pressure; ρ , liquid density; β , thermal expansion coefficient; ν , kinematic viscosity; $Pr = \nu/a$, Prandtl number; $Ra = (g\beta\Delta TH^3)/(\nu a)$, Rayleigh number; q_w , local heat flux; λ , thermal conductivity coefficient; $Nu_{loc} = (q_w H)/(\lambda \Delta T)$, local dimensionless heat flux; Nu_v , vertical-wall-average dimensionless heat flux; Nu_h , horizontal-wall-average dimensionless heat flux; $\dot{\mathbf{S}}$, strain rate tensor of averaged motion; C_f , friction coefficient; ν_t , turbulent viscosity; Pr_t , turbulent Prandtl number; k , kinetic turbulent energy; $\langle \tau_w \rangle$, time average wall friction stress; n , distance to the

nearest wall; $Y^+ = (\langle \tau_w \rangle / \rho)^{1/2} n_1 / \nu$, dimensionless coordinate of the centers of computational grid cells closest to the wall; Δ_i , linear dimension of computational cell in the direction of X_i .

ACKNOWLEDGMENTS

This study was supported by the Russian Program for support of leading scientific schools (grant NSh-1389.2003.8) and by the Russian Foundation for Basic Research (grant 05-02-17189a).

REFERENCES

1. Versteegh, T.A. and Nieuwstadt, F.T.M., *Int. J. Heat Fluid Flow*, 1997, vol. 19, p. 135.
2. Boudjemadi, R., Maupu, V., Laurence, D., and Le Quere, P., Direct Numerical Simulation of Natural Convection in a Vertical Channel: A Tool for Second-Moment Closure Modelling, in *Proc. Engineering Turbulence Modelling and Experiments 3*, Amsterdam: Elsevier, 1996, p. 39.
3. Peng, S.-H. and Davidson, L., *Int. J. Heat Fluid Flow*, 2001, vol. 22, p. 323.
4. Cabot, W. and Moin, P., *Flow Turbulence Combust.*, 1999, vol. 63, p. 269.
5. Spalart, P.R., *Int. J. Heat Fluid Flow*, 2000, vol. 21, p. 252.
6. Strelets, M., Detached Eddy Simulation of Massively Separated Flows, *AIAA Paper 2001-0879*, 2001.
7. Strelets, M.Kh., Travin, A.K., Shur, M.L., and Spalart, P.R., *Nauchno Tekh. Vedomosti SPbGPU (St. Petersburg State Polytech. Univ.)*, 2004, no. 2 (36), p. 22.
8. Nikitin, N.V., Nicoud, F., Wasistho, B., *et al.*, *Phys. Fluids*, 2000, vol. 12(7), p. 1629.
9. Smirnov, E.M., Recent Advances in Numerical Simulation of 3D Unsteady Convection Controlled by Buoyancy and Rotation, in *Proc. 12th Int. Heat Transfer Conf.*, Grenoble, 2002 (CD-ROM proceedings).
10. Abramov, A.G., The Method of Large Eddy Simulation in Application to Problems of Turbulent Convection in Volumes Heated from below: Options and Possibilities, *Cand. Sci. (Phys.-Math.) Dissertation*, St. Petersburg: St. Petersburg State Polytechnical Univ., 2003.
11. Abramov, A.G., Ivanov, N.G., and Smirnov, E.M., Numerical Study of High-Ra Rayleigh-Benard Mercury and Water Convection in Confined Enclosures Using a Hybrid RANS/LES Technique, in *Proc. Eurotherm Seminar 74*, Eindhoven, 2003, p. 33.
12. Tian, Y.S. and Karayiannis, T.G., *Int. J. Heat Mass Transfer*, 2000, vol. 43, p. 849.
13. Wolfshtein, M., *Int. J. Heat Mass Transfer*, 1969, vol. 12, p. 301.
14. Smirnov, E.M. and Zaitsev, D.K., *Nauchno Tekh. Vedomosti SPbGPU (St. Petersburg State Polytech. Univ.)*, 2004, no. 2 (36), p. 70.
15. Smirnov, E.M., Abramov, A.G., Ivanov, N.G., *et al.*, DNS and RANS/LES-Computations of Complex Geometry Flows Using a Parallel Multiblock Finite-Volume Code, in *Parallel Computational Fluid Dynamics. Advanced Numerical Methods Software and Applications*, Chetverushkin, B. *et al.*, Eds., Elsevier, 2004, p. 219.
16. Zimin, V.D. and Frik, P.G., *Turbulentnaya konveksiya (Turbulent Convection)*, Moscow: Nauka, 1988.