NUMERICAL INVESTIGATION OF HEAT TRANSFER UNDER LAMINARIZATION CONDITIONS OF TURBULENT FLOWS

V. G. Zubkov

UDC 532.5:533.6

Based on the mathematical boundary layer model, a numerical investigation of heat transfer is carried out for a wide range of turbulent Reynolds numbers in nozzles of experimental devices under laminarization conditions of turbulent flows.

Introduction. The experimentally established [1, 2] laminarization effects of turbulent flows in thermoenergetic devices have a quite practical value. Account of the variations in the structure of turbulent flows under the action of flow acceleration makes it possible to reach a better basis to designing real structures, which accelerates, in turn, the finishing of new products and reduces the amount of expensive tests. for example, for external flow, for gas flow between turbine blades, and in nozzle blocks the laminarization of turbulent flows leads to a positive effect, since it can assist in reducing the heat transfer intensity between the heated gas flow and the structural surface. A similar effect leads to undesirable results in heat transfer apparatus, where the heat transfer intensity must be largest. To account for flow features in the action zone of an accelerating flow in calculating heat transfer parameters in thermoenergetic devices it is necessary to use a sufficiently reliable method, capable of describing laminarization effects.

The boundary layer plays a fundamental role in hydrodynamic and thermal flow interactions with the surrounded surface. Therefore, a method of calculating heat transfer parameters in thermoenergetic devices, based on the mathematical boundary layer model, and valid for a wide range of turbulent Reynolds numbers [3], is suggested in the present paper. The boundary layer is assumed to be stationary, and the effect of mass forces and volume sources on heat and momentum transfer processes is not taken into account. The modified e–ε turbulence model is used for closure of the boundary layer equations. Its substantial differences from the basic model [4] have been analyzed in [3, 5]. Also presented are results implying the possibilities of the model in describing turbulent exchange for the various flow regimes. In that case the Reynolds number varied from $Re_x = 2 \cdot 10^4$ to $Re_x = 10^7$, while the turbulent Reynolds numbers R_t varied from 0 to 10^5. In particular, by means of the model it was possible to describe satisfactorily the laminar, transition, and turbulent flow regimes [3].

Direct solution of the boundary layer equations makes it possible to take into account local flow features, explain the role of the preceding boundary layer history, and establish the connection between the flow parameters in a direction perpendicular to the wall, keeping in mind the temperature dependence of the properties of the effective body, and pressure. In that case the flow outside the boundary layer is treated as one-dimensional without including the boundary layer effect.

1. Mathematical Boundary Layer Model. According to [5] the equations of the suggested boundary layer model for compressible fluid flow can be written in the following generalized form:

$$\frac{\partial \rho U \Phi}{\partial x} + \frac{\partial \rho V \Phi}{\partial y} = \frac{\partial}{\partial y}\left[\Gamma_d \frac{\partial \Phi}{\partial y}\right] + S_d, \qquad (1)$$

where Φ denotes the corresponding dependent variable in Table 1. (Also provided in the table are the closure relations and boundary conditions for the problem considered).

A mathematical model making it possible to calculate boundary layer turbulent flows must take into account two factors of different nature: the effect of small Reynolds numbers,

Moscow Institute of Automobile Construction. Translated from Inzhenerno-fizicheskii Zhurnal, Vol. 61, No. 2, pp. 223-229, August, 1991. Original article submitted March 27, 1990.

 0022-0841/91/6102-0976$12.50

TABLE 1. Variable Values in Eqs. (1), Closure Relations, and Boundary Conditions

Equation	Φ	Γ_d	S_d
Continuity	1	0	0
Motion	U	$\mu+\mu_t$	$-dp/dx$
Energy	T	$\frac{\mu}{Pr}+\frac{\mu_t}{Pr_t}$	$\frac{1}{C_p}\left[\mu\left(\frac{\partial U}{\partial y}\right)^2+\rho\varepsilon+Udp/dx\right]$
Intensity of turbulence	e	$\mu+\frac{\mu_t}{\sigma_e}$	$\mu_t\left(\frac{\partial U}{\partial y}\right)^2-\rho\varepsilon$
Turbulent energy dissipation	ε	$\mu+\frac{\mu_T}{\sigma_\varepsilon}$	$C_1\mu_t\frac{\varepsilon}{e}\left(\frac{\partial U}{\partial y}\right)^2-C_2f\rho\varepsilon^2/e$

Closure relations

$$\mu_T=C_\mu\mu R_t,\ R_t=e^2/(\nu\varepsilon),\ Pr_t=0{,}9,\ \sigma_e=1{,}0,\ \sigma_\varepsilon=1{,}3,$$

$$C_1=1{,}65,\ C_2=2[1-0{,}3\exp(-R_T^2)],$$

$$C_\mu=0{,}095[-\exp(-2{,}5)+\exp(-125/(50+R_t))],$$

$$f=-\exp(-10)+\exp(-250/(25+y_*^3)).$$

Boundary conditions

$$y=0,\ U=V=e=\varepsilon=0,\ T=T_{wa}(x)$$

$$y\to\infty,\ \partial U/\partial y=\partial T/\partial y=\partial e/\partial y=\partial\varepsilon/\partial y=0.$$

and the effect of wall proximity. The first effect is related to the dominating effect of molecular viscosity on the flow structure in direct vicinity to the surrounded surface. The second is due to preferential damping of velocity fluctuations along the normal to the wall. In the basic e–ε model [4] the closure relations are only functions of R_t, therefore the effect of the second factor on the fluctuating components is not included in it. To take into account the effect of wall proximity in the model suggested, we use an additional function f, depending on the dimensionless distance from the wall y_*, and including the dissipation equation in the last term of the right hand side as a factor. Its range of variation is from 0 to 1. The specific shape of the function f was determined as a result of a numerical experiment from the condition of guaranteeing best coincidence of mean velocity profiles and turbulent intensity with experimental data.

The system of equations (1), together with the closure relations, the boundary conditions, the equation of state, and the temperature and pressure dependences of the thermophysical medium parameters is solved by computer. The solution over the whole thickness of the boundary layer is implenented by a single algorithm directly from the surrounded surface to the external flow region. The original equations and boundary conditions are approximated by a standard implicit finite-difference scheme. The system of equations obtained in that case is solved by the steepest descent method. A modified coordinate system, guaranteeing "compression" of the transverse coordinate near the wall, is introduced so as to accelerate the computational process and guarantee the required accuracy. The conditions required to start the calculation are given by experimental and theoretical data for the laminar or turbulent flow regions [6].

2. Account of Compressibility of Effective Body. For high flow velocities it is necessary to take into account gas compressibility effects, leading to an increase in kinetic energy dissipation with heat. Consequently, the friction at the wall decreases in comparison with incompressible fluid flow. A numerical experiment was carried out to verify the dissipation mechanism adopted in the suggested model. The calculations were carried out for the case of a planar plate surrounded by a vanishing gradient air flow in the absence of heat transfer. Figure 1 shows the results of a calculation, handled in the form of a Mach number dependence of the ratio of the friction coefficient at the wall C_f during compressible flow to the corresponding value in an incompressible fluid. It follows from the figure that the model truly accounts for compressibility effects up to Mach numbers equal to 5. The deviation noted between the calculated and experimental data [7], observed for M > 5, can be explained

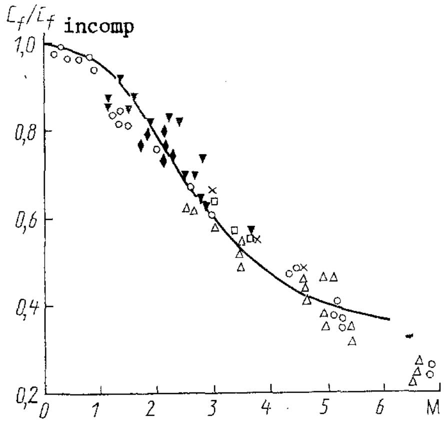

Fig. 1. The friction coefficient of a longitudinally streamlined plate as a function of the Mach number in a turbulent boundary layer: the points are experimental data [9], and the solid line is the model calculation.

by the simplifications adopted in constructing the mathematical model of turbulence, particularly in deriving the Reynolds equations, as well as by the fact that compressibility effects inside the fluctuating motion have not been accounted for. At high flow velocities the Mach number of turbulent transport increases, and the assumption of a negligibly small effect of density fluctuations on the flow parameters becomes invalid. The data obtained imply that the model can be used to calculate compressible flows in real thermoenergetic devices if $M < 5$.

3. Effect of Computational Grid Parameters on Calculation Results. Attempts of describing heat transfer in nozzles of thermoenergetic devices by means of empirical methods based on similarity theory, and by means of integral methods under conditions of large negative pressure gradients, do not provide agreement between calculation results and experimental data [2]. Our studies have not led to success, nor have attempts using models based on hypotheses of mixed path lengths or using one kinetic energy equation of turbulent motions [8], despite the fact that the calculations were carried out for nearly sonic flows. In that case, however, was noted a decisive effect of calculated grid parameters on the calculated values of flow characteristics. This effect is clearly shown in Fig. 2, where mean velocity profiles are shown in direct proximity to the surrounded surface. The points denote data, obtained by a calculation by model [8] for various flow velocity regimes for three different grids: $n = 40$, $n = 60$, $n = 120$. In the vanishing gradient flow regime ($K = 0$) the calculated profiles coincide with each other and with experimental data [9]. In that case at least three points are located within the limits of the laminar sublayer $y_* < 7$. In the case of flow acceleration under otherwise equal conditions a calculation by model [8] provides quite different velocity distributions in the boundary layer for each computational grid. The smaller the number of grid nodes n, the further the first computational point is located

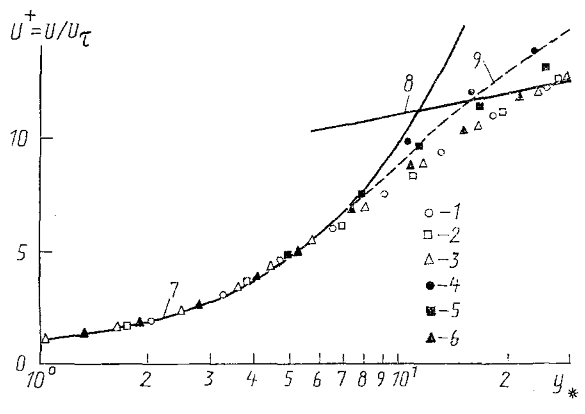

Fig. 2. Effect of computational grid parameters on modeling results of velocity profiles in the boundary layer region: points — calculation by model [8] for different values of the acceleration parameter and computational grid parameters; 1) $n = 40$ and $K = 0$; 2) $n = 60$ and $K = 0$; 3) $n = 120$ and $K = 0$; 4) $n = 40$ and $K > K_{cr}$; 5) $n = 60$ and $K > K_{cr}$; 6) $n = 120$ and $K > K_{cr}$; 7) $U^+ = y_*$; 8) $U^+ = 5.5\ \lg y_* + 5.45$; 9) calculation by model [5] for $n = 120$, $K = 2.2 \cdot 10^{-6}$, $Re_x = 1.5 \cdot 10^6$.

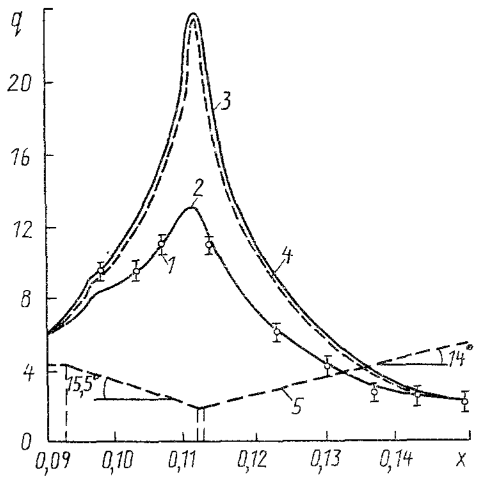

Fig. 3. Effect on computational grid parameters on results of modeling heat transfer in nozzles of experimental devices: 1) experimental data [2]; 2) calculation by model [8] for n = 40; 3) by model [8] for n = 120; 4) by the method of [10]; 5) nozzle contour, q in MW/m^2, x in m.

from the surrounded surface, and the more substantial is the deviation of the velocity profile from the universal logarithmic law. An artificial laminarization of the velocity profile occurs, generating corresponding variations in the flow parameters, particularly the tangential stress of friction at the wall and specific thermal flux from the heated gas to the surrounded surface.

In Fig. 3 we mark by points experimental data on heat transfer in a nozzle [2], compared with calculation results by model [8] for two n values. As seen from the data provided, the use of n = 40 due to artificial laminarization leads to agreement between calculated and experimental data. For n = 120 the velocity profiles in Fig. 2 are not deformed, but neither is there agreement between calculated values of the thermal flux (curve 3) and the experimental data in Fig. 3. It must be noted that this curve is in good agreement with the calculation results of thermal flux by method [10], obtained in [2] (curve 4). In both cases the result exceeds the experimental data in the zone of maximum values by a factor of two.

Since the effect noted of artificial laminarization of the mean velocity profiles is caused by the choice of the computational grid, it can be avoided by selecting conditions such that for any extent of flow acceleration the first three solution points are located in the region $y_* < 5$. I has been established [5] that the modified e–ε model takes into account flow features in the boundary layer, including in the presence of flow acceleration. This too, though to a lesser extent, is subject to the effect of a difference grid. In an accelerated flow ($K > K_{cr}$) at n = 40 the calculation by the e–ε model practically coincides with the corresponding data for model [8], represented in Fig. 2. For n = 120, $K = 2.2 \cdot 10^{-6}$ and $Re_x = 1.5 \cdot 10^6$ (curve 9) the calculation by the suggested model is in full agreement with the experimental data of [4]. Further increase of the node number of the computational grid (under otherwise unchanged calculation parameters) does not lead to a change in this result, therefore all subsequent calculations for accelerated flows are carried out for n = 120.

4. Calculation of Heat Transfer under Laminarization Conditions of Turbulent Flows. As an example of the possibilities of the suggested model we show in Fig. 4 results of heat transfer calculations in nozzles of an experimental device [2]. The calculation was carried out within the planar boundary layer approximation, since under the conditions of the problem considered the thickness of the boundary layer is substantially smaller than the radii of curvature of the nozzle surface. As boundary conditions we used the wall temperature values T_{wa} and the mean pressure for each series of experiments.

Under conditions of high temperatures and effective bodies of complex composition, processes such as diffusion, chemical reactions, and two-phase flows play a substantial role in boundary layers of thermoenergetic devices. The features of these processes can be accounted for following determination of the effective body composition. The effective body composition was calculated by the program of [11]. The results of this solution are introduced in the model in the form of the temperature and pressure dependences of the thermophysical properties of the gas.

The results of heat transfer calculations are presented for two types of conical nozzles, differing from each other by the opening angles in the subcritical and supercritical regions.

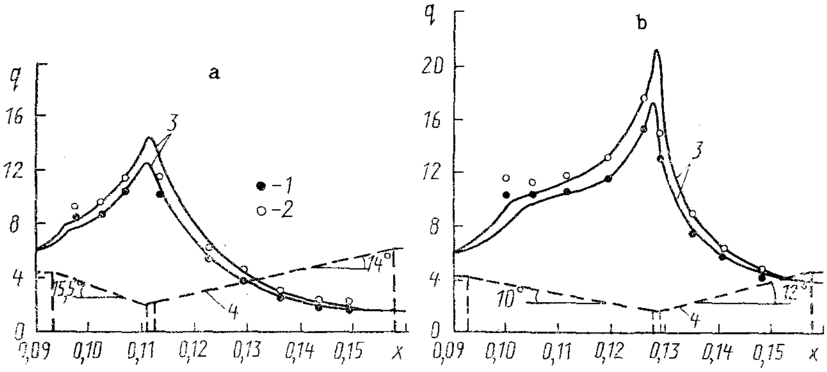

Fig. 4. Heat transfer calculation in nozzles of an experimental device: a) results for nozzle No. 1; b) for nozzle No. 2; points are experimental data [2] for two moments of time; 1) $\tau = 1$ sec; 2) 2 sec; 3) model calculation for corresponding moments of time; 4) nozzle contour.

This configuration choice of the internal contour guaranteed reaching values of the acceleration parameter $K \sim 5 \cdot 10^{-6}$ and $2.5 \cdot 10^{-6}$, respectively, for nozzles No. 1 and No. 2 in the subsonic portion. As follows from the plots, the model suggested truly describes the experimental data.

The satisfactory agreement between experimental data on heat transfer in nozzles, obtained under nonstationary conditions, with calculations by stationary methodology is explained by the fact that under the flow conditions considered the effect of the nonstationary parameter is negligibly small. It is well known that the larger the ratio of the characteristic length to the product of the characteristic velocity and time (the Strouhal number), the more strongly the effects related to nonstationarity must be manifested. Analysis of the calculation results for Strouhal numbers 0, 1 has shown [12] that the divergence between the solutions of the stationary and nonstationary problems does not exceed, under these conditions, 1% in the thermal flux calculated at the wall. The experimental data [2] were obtained for Strouhal numbers not exceeding 0, 1, therefore nonstationary effects are small, and calculations by the stationary methodology are justified. This is confirmed by the results of Fig. 4. The thermal flux behind nozzle 1 varies by less than 10-15%.

Conclusions. A method was suggested for calculating heat transfer under laminarization conditions of turbulent flow in thermoenergetic devices. The method is based on the mathematical boundary layer model for a wide range of turbulent Reynolds numbers. It has been shown that the method truly accounts for medium compressibility effects up to Mach numbers equal to 5. A substantial effect of computational grid parameters on the results obtained was established in the direct vicinity of the surrounded surface, and their optimal values were selected. Calculations were carried out of heat transfer in nozzle blocks of experimental devices. The method suggested describes correctly flow features in the laminarization region of turbulent flows under the action of an accelerated flow, and can be recommended as an engineering tool in calculating flow parameters in thermoenergetic devices.

NOTATION

Here x denotes the coordinate along the surface; y is a coordinate perpendicular to the surface; U is the velocity component along the x-axis; V is the velocity component along the y-axis; p is pressure; T is temperature; u_i' are velocity fluctuation components, with $i = 1, 2, 3$; $e = 1/2 \sum_{i=1}^{3} \langle (u_i')^2 \rangle$ is the turbulence intensity; $\varepsilon = \nu \sum_{ij=1}^{3} \langle (\partial u_i'/\partial x_j)^2 \rangle$ is the isotropic part of the total turbulent energy dissipation; μ is the dynamic viscosity; ν is the kinematic viscosity; ρ is the density; C_p is the fluid heat capacity at constant pressure; Φ is a dependent variable, Γ_d is the diffusion coefficient, S_d is the source term; μ_t is the turbulent exchange coefficient; R_t is the turbulent Reynolds number; Pr_t is the turbulent Prandtl number; C_1, C_2, C_μ, σ_e, σ_ε are coefficients; f is a correction function in the dissipation equation, $y_* = yU_t/\nu$ is a dimensionless coordinate perpendicular to the surface; $U_\tau = \sqrt{\tau_{wa}/\rho_{wa}}$ is the dynamic velocity; τ_{wa} is the friction stress on the surrounded surface;

$K = (\nu/U_\infty^2)\cdot(dU_\infty/dx)$ is the acceleration parameter ($K_{cr} = 2.5\cdot10^{-6}$); q is the thermal flux; T_{wa} is the temperature of the surrounded surface; U_∞ is the velocity of the incoming flow; n is the number of nodes of the computational grid in the direction of the y-axis; M is the Mach number; Re_x is the Reynolds number over the x-coordinate; and C_f denotes the local friction coefficient.

LITERATURE CITED

1. L. G. Back, R. F. Kaffel, and P. F. Massey, Teploperedacha, 92, No. 3, 29-40 (1970).
2. V. G. Zubkov, Determination of Thermal Fluxes in Model Nozzles, VINITI October 27, 1989, No. 6553-V89, Moscow (1989).
3. V. G. Zubkov, Inzh.-fiz. Zh., 48, No. 5, 746-754 (1985).
4. W. P. Jones and B. E. Launder, Int. J. Heat Mass Transf., 15, Pt. 2, 301-314 (1972).
5. V. G. Zubkov, Zh. Prikl. Mekh. Tekh. Fiz., No. 2, 71-78 (1985).
6. J. O. Hinze, Turbulence, 2nd ed., McGraw-Hill, New York (1975).
7. Yu. V. Lapin, Turbulent Boundary Layer in Supersonic Gas Flows [in Russian], Moscow (1982).
8. G. S. Glushko, Izv. Akad. Nauk SSSR, Mekh., No. 4, 13-23 (1965).
9. H. Schlichting, Boundary-Layer Theory, 7th ed., McGraw-Hill, New York (1979).
10. M. V. Dobrovol'skii, Fluid Rocket Engines [in Russian], Moscow (1968).
11. G. B. Sinyarev, B. G. Trusov, and L. E. Slyn'ko, Tr. Mosk. Vyssh. Tekh. Uchil., No. 159, 60-71, Moscow (1973).
12. V. M. Paskonov, V. I. Polezhaev, and L. A. Chudov, Numerical Simulation of Heat and Mass Transfer Processes [in Russian], Moscow (1984).

HEAT TRANSFER BY MIXED CONVECTION IN A MOVING ROD BUNDLE

V. I. Eliseev and Yu. P. Sovit

UDC 532.54:536.24

A mathematical model of mixed convection in moving rod bundles is proposed, and cooling in an open space is analyzed. Estimates of the local Nusselt numbers are obtained for quasistabilized conditions.

The formation of fibers from polymer melts is fundamental to the production of synthetic fibers. The physical properties and quality of the fibers depend significantly on the heat-transfer intensity of the moving polymer jets with the surrounding medium. The extension of single fibers has now been fairly completely studied, and the corresponding mathematical models have been constructed [1, 2]. However, it is more common to form bundles of fibers. In such conditions, the spatial interaction of individual fibers and of the whole bundle with the external medium must be considered. The few studies of heat transfer in such conditions have mainly been qualitative and experimental in character [3, 4]. Accordingly, it is necessary to develop mathematical models and methods of calculation of heat transfer in moving fiber bundles. In addition, it is of interest to determine the role of various physical factors in the overall pattern of heat transfer of the fiber bundle with the external medium. Thus, in the case of low velocities and high melt temperature (glass-fiber), free convection is important. Increase in the rate of fiber formation leads to increase in the role of induced convection. It is known [1] that, in the existing conditions of formation of a single synthetic fiber, the proportion of free convection in the overall heat-transfer process is around 10%. As yet, there are no such estimates for bundles of fibers.

The filtrational flow model is widely used for the description of heat transfer in complex rod systems [5, 6]. It reflects the hydrodynamic and thermal interaction of the rods with one another and of the bundle as a whole with the surrounding atmosphere. In the literature, attention focuses on power units with high gas velocities, and accordingly

Dnepropetrovsk State University. Translated from Inzhenerno-fizicheskii Zhurnal, Vol. 61, No. 2, pp. 230-238, August, 1991. Original article submitted April 15, 1990.