Moreover, negative pressure, increasing toward the axis of the tornado, acts to draw the nearly untwisted near-surface boundary layer of the wind flow into the central jet. Dust and water droplets also reduce angular velocity. These factors explain the above-noted absence of twisting in the ascending central jet. In experiments, R. S. Trofimov observed a vortex with a nonrotating core in a conical model chamber (the core was visualized by axially introducing bromine vapors into the chamber).

In conclusion, we noted that the existence of high pressure (in the zone where the annular jet meets the surface) and low pressure (in the central jet) is supported by the character of failures which occur in tornadoes (for example, the roofs of buildings are not only pushed in, but are also torn off and carried some distance). A cross wind distorts the axis of a tornado, as is evident from Fig. 5. The flow pattern inside a tornado described here is fairly stable, which helps to conserve it as the tornado is transported tens of kilometers by the wind flow.

NOTATION

u_a, u_t, translational and circumferential components of velocity of the gas flow; V_1, volume flow rate of the gas; Γ, gas velocity circulation; p, gas pressure; r, radius of the cross section.

LITERATURE CITED

1. G. N. Abramovich, Theory of Turbulent Jets [in Russian], Moscow (1960).
2. G. N. Abramovich, T. A. Girshovich, S. Yu. Krasheninnikov, et al., Theory of Turbulent Jets [in Russian], 2nd ed., Moscow (1984).
3. F. K. Gouldin, S. J. De Bakey, and S. L. Li, Aerosp. Technol., No. 10, 46-56 (1985).
4. D. I. Raleigh and H. T. Somerset, Aerosp. Technol., No. 10, 56-66 (1985).
5. Mixing-Type Heater, Inventor's Certificate No. 338766 USSR: MKI F 28c3/06, F 22d1/44.
6. Yu. F. Dityakin, L. A. Klyachko, B. V. Novikov, et al., Atomization of Liquids [in Russian], Moscow (1977).
7. O. I. Navoznov, A. A. Pavel'ev, A. S. Mul'gi, and M. K. Laats, Turbulent Two-Phase Flows [in Russian], Tallin (1979), pp. 158-165.
8. A. I. Mikhailov, Yu. M. Gorbunov, V. B. Rutovskii, et al., Operation and Design of Combustion Chambers in Gas-Turbine Engines [in Russian], Moscow (1956).

NUMERICAL MODELING OF COMPLEX VISCOUS FLUID VORTICAL FLOWS

I. A. Belov

UDC 532.517.2:4

The influence of the turbulence model, numerical diffusion, and methods of formulating the boundary conditions on a solid wall on computation results is analyzed in an example of a finite-difference simulation of viscous fluid vortical flows in a broad range of Mach and Reynolds number variation.

Despite the success in applying electronic computers to solve different fluid mechanics and heat and mass transfer problems, the relation to the information, obtained as a result of such a computational experiment, for potential consumer-specialists in the design of new engineering is still extremely cautious if not generally negative in many cases. To a considerable extent this is explained by the designers of engineering not being prepared to perceive the idea of a computational experiment, which is due not only to the majority not having experience in working by using an electronic computer but also to the constraints of purely computational nature on the applicability of the computation results, which are governed by features of the numerical procedures constructed, the turbulence models selected, the boundary conditions formulated, etc.

In this paper, the available relationships of the possible defects and difficulties of reproduction of real flow properties in the computational experiment to the mentioned cons-

---

Civil Aviation Academy, Leningrad. Translated from Inzhenerno-Fizicheskii Zhurnal, Vol. 53, No. 5, pp. 758-765, November, 1987. Original article submitted April 3, 1987.

0022-0841/87/5305-1266 $12.50

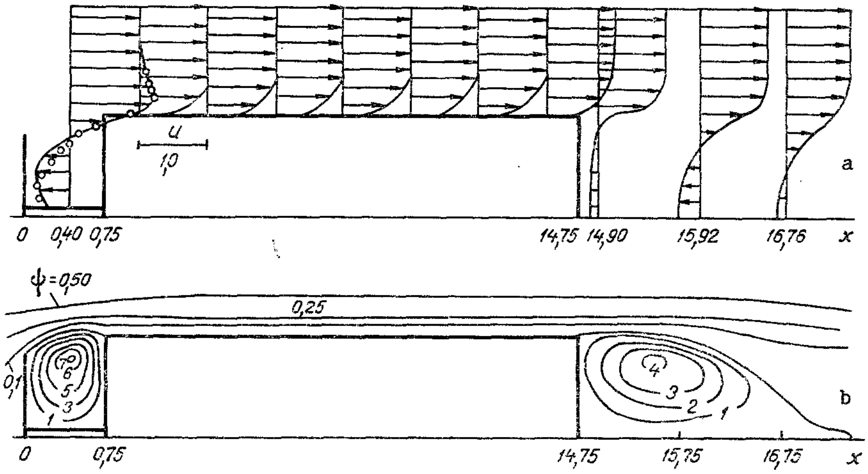

Fig. 1. Subsonic flow around a cylinder with a disc. Profiles of the longitudinal velocity component (a) and the stream line ψ = const (b): 1) $\psi = -0.01$; 2) (-0.02); 3) (-0.03); 4) (-0.04); 5) (-0.06); 6) (-0.09); 7) (-0.1); the points are from experiment [3].

traints are discussed in an example of numerical simulation of viscous flows. The simulation is based on a finite-difference representation of the initial differential equations written in the Navier–Stokes or Reynolds form in the majority of cases for a broad range of variation of the Mach and Reynolds numbers. Research performed by the author and his colleagues is drawn upon as illustrative material. Let us note the common distinction for all the research utilized, which is the investigation of flows with developed circulation zones.

We start with the selection of the turbulence model. Models of differential type on a par with phenomenological models are used in a semiempirical theory of turbulence, and in addition to equations for the average stream parameters equations for the turbulence characteristics are also introduced in the considerations (see [1], say). Their application is not accidental: in complex flows accompanied by stream separation, by heat transfer, three-dimensionality, and other phenomena difficult to subject to a theoretical analysis, apparently only those turbulence models more complex than the phenomenological can permit access to the solution of problems of practical importance. Meanwhile, these models cannot be considered as the means for expanding our knowledge about the nature of turbulence since modeling, as such, does not disclose any new properties although it permits analysis of the mechanisms controlling turbulence in the majority of cases. Thus, the construction of multiparametric models utilizing the equations for the Reynolds stress tensor components affords a possibility of estimating the functional structure of turbulence in mutual interactions of each component. However the introduction of a large number of equations for the composite elements of turbulence requires appropriate empirical information, which the more difficult it is to obtain experimentally, the more complex the model. Taking account of the level of development of modern experimental apparatus, this last fact speaks more often about the difficulties of constructing turbulence models with a large number of differential equations rather than about their potential possibilities at this time. The time expenditure in a numerical solution of a large number of differential equations also hinders the application of complex multiparameter turbulence models. In this connection, estimating the promise of the direction of a semiempirical turbulence theory including multiparametric model development as a whole, even simpler models, including the phenomenological, should not be neglected, especially in those cases when sufficient experimental information about the flows under investigation is available.

The computational and experimental data on the flow configuration near a cylinder surface with a disc of radius r mounted in front at a distance ℓ [2], as represented in Fig. 1, are illustrations of the possibilities of differential turbulence models. The computation is performed under the assumption of fluid incompressibility ($M = 0$) at a Reynolds number defined by the unperturbed stream velocity and the cylinder radius $Re = 10^5$. In the computation $\ell = r = 0.75$; the cylinder elongation is $L = 14$ (in fractions of the cylinder radius). In addition to the continuity and Reynolds equations, the initial system of equations included also an equation for a two-parameter dissipative model of turbulence $k-\varepsilon$ in a so-

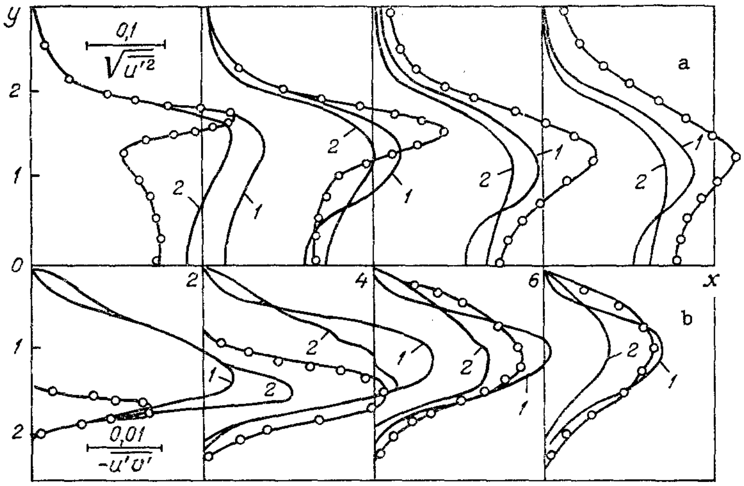

Fig. 2. Flow in the wake behind a disc. Profiles of the r.m.s. fluctuating velocity component (a) and the shear stress (b): 1) Reynolds stress model; 2) dissipative k − ε model in a high Reynolds approximation (in Re_T); points are from experiment [4].

called high Reynolds approximation in the magnitude of the turbulent Reynolds number ($Re_T \to \infty$). Comparision of the computed and experimental data [3] permits making a conclusion about the sufficiently accurate description of the complex vortex configuration both ahead of the cylinder and in the wake behind it. The fact of essential effect of external stream turbulence on the vortex flow is established here (in this example Tu = 0.5%). In particular, its presence diminishes the degree of rarefaction in the stream between the disc and the cylinder and results in growth of the frontal drag of the component under investigation. This latter circumstance must be taken into account in transferring the results of a wind tunnel experiment to full-scale objects.

Taking Tu ≈ 0 in the next examples, we compare turbulence models of differential type utilized in a computation of the near wake behind a disc for M = 0 and $Re = 3.5 \cdot 10^4$ (the same characteristic quantities are selected here as in the previous case). A dissipative k − ε turbulence model in a high Reynolds approximation (in Re_T) and an algebraic variation of a multiparametric model of Reynolds stresses obtained within the framework of an assumption about the proportionality of the transfer of the Reynolds stresses and the energy k are compared (see [1], say). Presented in Fig. 2 are results of a computation in the form of profiles $\sqrt{\overline{u'^2}}\,(y)$; $\overline{u'v'}(y)$ for different sections x behind the disc. Experimental data taken from [4] are also presented in the figure. Attention is turned to the repeatability of the results obtained by using different turbulence models, however the best agreement between the computation and experiment results is observed for the Reynolds stress model especially for short ranges from the disc surface.

A finite-difference scheme [5] with the convective terms in the equations approximated by quadratic differences opposite to the stream direction is used in the previous examples. Following [5], where it is shown that the solution for the characteristics k and ε is less sensitive to the method of discretizing the initial equations, especially in the domain of a strong shift to the boundary of the separation region, the finite-difference analogs of the differential equations for k and ε are constructed by the so-called "hybrid" scheme that combines central differences and unilateral differences opposite to the stream direction. Outside the dependence on the turbulence model, a method of pressure correction is used in the numerical procedure which is substantially a relaxation analog of the known method of splitting according to physical processes. The difference equations obtained are solved by a linear scanning method. Let us note that because of the developed nature of the flow, due to the disruption of the stream at the sharp edges of the body, the stream parameters and turbulence characteristics in the cells of the difference mesh adjacent to the walls are determined by using the method of near-wall functions constructed under the assumption of local equilibrium of the turbulent pulsation energy and local isotropy of the dissipating vortices in the near-wall domain (see [1, 5], for instance). The conditions on the remaining boundaries of the computation domain are formulated in the traditional manner.

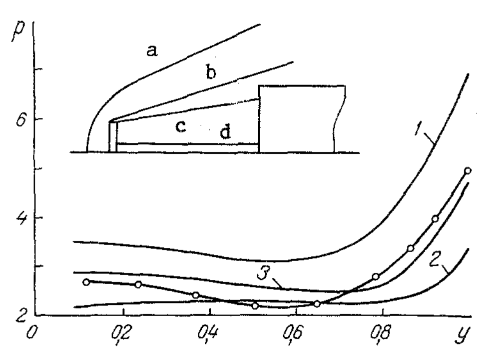

Fig. 3

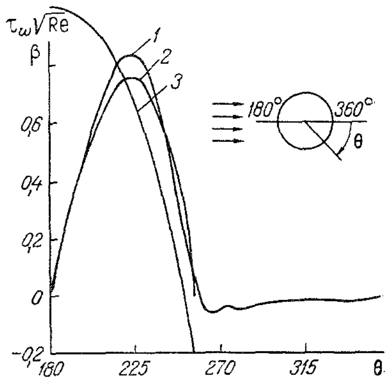

Fig. 4

Fig. 3. Supersonic flow around a cylinder with a disc [fundamental structural elements: a) bow shock; b) shock layer; c) turbulent mixing layer; d) separation zone]. Static pressure profiles on the cylinder endface surface: 1) computation of a rectangular mesh; 2) on an oblique-angled mesh; 3) on an oblique-angled mesh with superposed mixing layer; points are from experiment.

Fig. 4. Flow around an isolated cylinder: 1) computation of $\tau_w\sqrt{Re}$ from the profile of the longitudinal velocity component in the near-wall layer; 2) computation of $\tau_w\sqrt{Re}$ by the local similarity method: 3) computation of β.

Selection of the scheme, the method of solving the difference equations, the method of formulating the boundary conditions and other "schematic" factors is an important component of the computational experiment. It is necessary to refer here to questions of constructing the difference networks, assuring the stability of the numerical procedure, taking account of the influence of numerical diffusion due to errors in discretization of the initial differential equations, and clarification of methods for its elimination, etc. Consequently, the calculator often arrives at a situation when these aspects of the computational experiment darken or completely displace the physical comprehension of the problem under consideration. The following example [6] indicates the inadmissibility of such a formalization of the computational experiment.

Supersonic flow is examined around a body of the same cylinder-disc configuration as is displayed in Fig. 1, for $M = 4.15$, $Re = 3.2 \cdot 10^6$, $\ell = 2.9$ and $r = 0.23$ (the cylinder radius and the unperturbed stream velocity are taken as characteristic dimensions). The computations are performed by using the Godunov discontinuity dissipation scheme of first order accuracy applied to a nonstationary system of Euler equations on two types of difference meshes: rectangular associated with the cylindrical coordinate system utilized, and oblique-angles with network lines oriented along the hypothetical direction of the mixing layer on the outer boundary of the circulation flow domain between the disc and the cylinder. As it turned out, the solution of the problem depends substantially on the type of computational mesh. In particular, a significant mismatch between the pressure profiles on the forward cylinder endface (see Fig. 3) is due to inaccuracy in the modelling in the computation of exchange processes between the external stream and the stream in the circulation zone (the pressure in the figure is referred to the corresponding quantity in the unperturbed stream). A deduction is made that the kind of mesh exerts a stronger influence on the solution than a change in the significant limits of the mesh spacing. Computations on a rectangular mesh showed the presence of intensive transfer in the mixing layer domain because of the action of the numerical diffusion mechanism that appeared in the formation of the developed circulation zone between the disc and the cylinder. An almost total exclusion of numerical diffusion in the mixing layer domain in computations on an oblique-angled mesh results in abrupt attenuation of the circulation flow. Apparently different solutions can be obtained for the construction of different difference meshes with an identical (bounded) number of cells, therefore the confidence in the solution of the problem under the conditions of the idealized formulation under consideration is not obvious. Introduction of a turbulent transport mechanism in the mixing layer permits elimination of this disadvantage of the computation even in the case of applying simple phenomenological turbulence models. A system of Reynolds and energy equations for stream parameters averaged with respect to time,

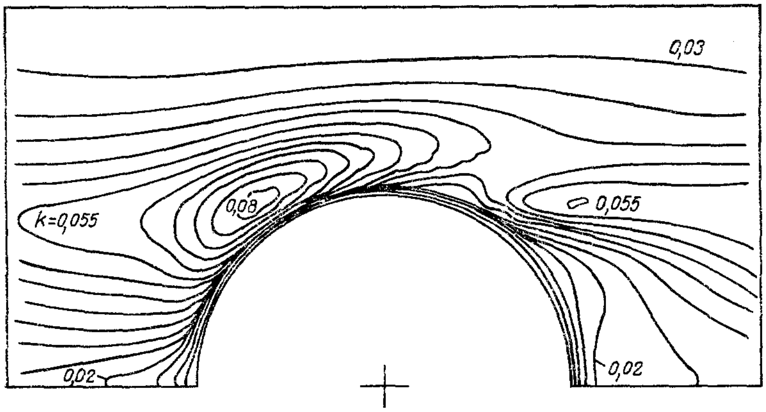

Fig. 5. Flow around a cylinder in an in-line tube bundle. Lines of constant values of the kinetic energy of the turbulent fluctuations (constructed with a spacing).

which are closed by using the semi-empirical convective Prandtl model, is used in [6] to describe turbulent fluid motion in the mixing layer.

Let us examine the influence of numerical diffusion on the computation results in more detail. Complete comprehension of the action of the numerical diffusion mechanism has not yet been achieved although researches are known in which attempts have been made to take it into account. In particular, the following reasoning à propos of this is presented in [7]: 1) numerical diffusion actually means that the mechanism of convective-diffusive exchange is distorted in the computational domain; 2) the magnitude of numerical diffusion depends on the method of writing the initial system of differential equations as well as the method of arranging the mesh nodes in the computational domain; 3) an increase in the order of the accuracy of the difference approximation of the initial equations does not always result in a diminution in the numerical diffusion (in this respect the form of the approximation of the convective terms is most important). Complete elimination of numerical diffusion when using modern electronic computers is impossible; the diminution of its influence on the results of a computation is assured because of the divergent mode of writing the initial equations; the difference approximation of convective terms by using a counter-flow scheme of a high order of accuracy, the Leonard scheme mentioned earlier, say; adaptation of the difference mesh to characteristic singularities of the real flow configuration, etc.

It should be noted that even for a correct selection of the turbulence model and assurance of a minimal magnitude of numerical diffusion in the computational domain, it is impossible to guarantee the confidence of the computational experiment results because of the possible errors in formulating the boundary conditions. The quantity of variations in giving conditions on different boundaries is sufficiently great, consequently, we here touch upon just the most difficult problem of formulating conditions on a solid wall. At first glance everything appears simple here. Indeed, for the flows under consideration there is no special difficulty in realizing the condition of fluid adherence to the wall. However, the necessity of an accurate computation of the friction τ_w on the wall imposes a constraint on the mesh spacing $h \ll \sqrt{Re}$. The possibilities of modern electronic computers in the majority of computations evidently do not permit making the mesh infinitely finer as the Re number grows. Consequently, it is necessary to rely on more complex methods of computing τ_w as compared with the method utilized for small Re numbers, which is based on an analysis of the near-wall profile of the longitudinal velocity component. The classical method of local similarity is proposed in [7, 8], for example, for these purposes in the numerical simulation of flows at high Re numbers in both the laminar and turbulent modes.

Application of the local similarity method will be illustrated in an example of analyzing the flow around an isolated circular cylinder by a uniform stream at $M = 0$ in the laminar range of Re number variation. Since the method is developed for the initial system of equations in the boundary layer approximation, we solve the appropriate equation for a certain given Falkner–Skan parameter β for each mesh cell of the computation domain adjacent to the wall. This latter is determined for the cell under consideration from the solution of the initial system of equations in Navier–Stokes form. The friction at the wall is consequently found and later utilized to compute the diffusion flow of the longitudinal velocity component in the near-wall cells to construct an algorithm of the solution of the Navier–Stokes equations in the whole computation domain.

Curves of the τ_w distribution obtained by different methods (normalized with respect to the magnitude of the free stream velocity head) on the cylinder surface [7] are constructed in Fig. 4 for $Re = 10^3$ determined according to the cylinder diameter and the unperturbed flow velocity. We note the possibility of a substantial increase in the upper bound of the Re number up to which the friction τ_w is computed exactly even on coarse meshes as an advantage of the method proposed for the formulation of the boundary conditions on a wall by using the local similarity method. The correct location of the stream separation point on the cylinder surface ($\beta = -0.1988$ in value) is achieved in the computation and, consequently, the correct reproduction of the remaining local and integral flow characteristics in the numerical simulation also.

The local similarity method is utilized in [7, 8] to compute turbulent flows also, when the initial system of equations is written in Reynolds form. However, the most widespread method of formulating boundary conditions on the wall in this case is the method mentioned earlier of near-wall functions that is ordinarily realizable within the framework of a high-Reynolds dissipative model of turbulence $k - \varepsilon$. Moreover, complicated versions of the near-wall functions method are also known, including for subsonic compressible fluid flows that permit taking account of the influence of the viscous sublayer on the computation of the friction and the turbulence characteristics k and ε in the near-wall layer more accurately. One of the effective approaches in this respect is applied in [9], in which computed data, that agree well with experiment, are obtained about the flow around a plate with installed transverse ribs.

It is sometimes not possible to yield the preference to some method of formulating the boundary conditions on the wall without taking account of the structural features of the flow under consideration. Thus, domains where application of the method of near-wall functions is not justified physically always exist, as a rule, in the separation mode of the flow around a body. Among them in particular are the neighborhood of the forward and rear stagnation points in the example considered above of the flow around a cylinder, and also the point of flow separation on a cylinder surface. In this case, errors in the computation when using the method of near-wall functions can distort substantially both the location of the flow separation point on the body and the configuration of its flow as a whole. Therefore, a careful check on the physical foundation of the assumptions taken is necessary when formulating the conditions on the wall. An example of a computation taken from [7], in which the structural features of the flow are taken into account, is illustrated in Fig. 5 where the change in the kinetic energy of the turbulent fluctuations is shown in the form of lines of constant values of k for an element of an in-line tube bundle at $Re = 10^4$ (the cylinder diameter is selected as characteristic dimension; and its average mass flow rate in the gap between the tube bundles is used as characteristic velocity). Here the fluid is taken incompressible when conducting a calculation experiment, and low-Reynolds $k - \varepsilon$ approximation of the turbulence model is used at the wall when the turbulent Reynolds number is $Re_T \to 0$. The boundary conditions on the wall are formulated with relationships of the local similarity method taken into account. As a singularity of the computation we note the utilization of periodic conditions at the input and output boundaries of the computation domain.

Summarizing, its practicality should especially be emphasized. Indeed, here the principal attention has been paid to problems of the compatibility between the numerical and physical modelling of complex vortex flows of a viscous fluid. A number of no less important questions of a calculational nature, such as the efficiency of the calculation procedures, the optimization of the mesh node distribution in the computational domain, and the mapping of the complex geometry, checkout, and processing of the numerical information obtained during the experiment, etc., are hardly touched upon here. Without examining these aspects of the calculational experiment since they are the subject of a special investigation, we indicate as the conclusion of the analysis performed that the calculational experiment is fully realizable for specific engineering problems despite the difficulties presently existent. Its advantages are evident in the search for new structural and technological solutions for apparatus and processes being developed. It is also indubitable that it will be applied in investigations of flows whose reproduction is difficult or impossible under laboratory conditions. In particular, significant achievements can be expected from the calculational experiment for the solution of problems of turbulent boundary layer control because of the change in its turbulence configuration. In this respect, the method considered in [10] of organizing near-wall turbulent flows by forming developed circulation zones at the surface of the streamlined body is promising. The results of computing the flow by a sub- and supersonic flow around a body with a forward separation zone, as displayed in Figs. 1 and 3, are an illustration of the positive properties of flows organized in such a manner.

NOTATION

x, y, coordinate along the stream and in a transverse direction; u, v, velocity components in the x and y directions, respectively; k, kinetic energy of the turbulent fluctuations; ε, rate of dissipation of k; ℓ, distance between the disc and the cylinder; r, disc radius; L, cylinder elongation; h, dimension of the difference mesh step; τ_w, friction on the wall; β, Falkner–Skan parameter; ψ, stream function; θ, an angular coordinate; M, Mach number; Re, Reynolds number; $Re_T = k^2/(\nu\varepsilon)$, turbulent Reynolds number; ν, kinematic viscosity coefficient; Tu, intensity of external flow turbulence, $(\ )'$, fluctuation; $(\bar{\ })$, time-average.

LITERATURE CITED

1. I. A. Belov, Turbulence Models (Textbook) [in Russian], Leningrad (1986).
2. V. K. Bobyshev and S. A. Isaev, Turbulent Transfer Processes [in Russian], Minsk (1987).
3. K. Koenig and A. Roshko, J. Fluid Mech., 156, 167-204 (1985).
4. T. Carmody, Theoretical Principles of Engineering Computations, 86, No. 4, 281-296 (1964).
5. B. P. Leonard, Comp. Meth. Appl. Mech. Eng., 19, No. 1, 59-98 (1979).
6. I. A. Belov, S. A. Isaev, and A. Yu. Mitin, Inzh.-Fiz. Zh., 51, No. 4, 555-563 (1986).
7. I. A. Belov and N. A. Kudryavtsev, Heat Elimination and Resistance of Tube Bundles [in Russian], Leningrad (1987).
8. I. A. Belov, Inzh.-Fiz. Zh., 51, No. 3, 395-402 (1986).
9. L. G. Artamonova, I. A. Belov, V. I. Mamchur, et al., Inzh.-Fiz. Zh., 52, No. 1, 43-51 (1987).
10. I. A. Belov, Interaction of Nonuniform Flows with Obstacles [in Russian], Leningrad (1983).

THEORY AND EXPERIMENT IN THE PROBLEM OF THE INTERACTION OF HIGH-VELOCITY GAS FLOWS WITH MATERIALS

Yu. V. Polezhaev

UDC 533.6.011.6

An analysis is made of the intensive thermal action of a gas flow on a body in the flow. An examination is made of features of failure of such bodies and the difference between the melting and vaporization points of glass-plastics in a turbulent boundary layer compared to a laminar boundary layer.

The interaction of high-velocity gas flows with the surfaces of different bodies has been intensively studied for about 30 years. During this period, significant strides have been made in studying different processes and phenomena in the boundary layer of the bodies, on the body surface, and inside the heated layer of material. Experimental studies have been vigorously pursued and numerous types of test stands have been developed. Some of these stands have a maximum power of tens of thousands of kilowatts.

The present article focuses on analysis of the intensive thermomechanical action of a gas flow on a body in the flow, which is usually referred to as the problem of thermal protection. Such terminology is arbitrary to a considerable extent, since the body is acted upon and brought to failure not only by heat flows, but also by diffusion flows of the chemically active components and by mechanical loads in the form of pressure and friction gradients. In contrast to most of the studies published recently, here we analyze processes occurring on the surface undergoing failure with a supersonic flow and a turbulent boundary layer.

Translated from Inzhenerno-Fizicheskii Zhurnal, Vol. 53, No. 5, pp. 765-774, November, 1987. Original article submitted February 3, 1987.

0022-0841/87/5305-1272 $12.50