DEVELOPMENT AND INVESTIGATION OF THE TECHNOLOGIES INVOLVING THERMAL PROTECTION OF SURFACES IMMERSED IN DISPERSE WORKING MEDIUM FLOW

V. N. Kovalnogov,* R. V. Fedorov, L. V. Khakhaleva, D. A. Generalov, & A. V. Chukalin

Ulyanovsk State Technical University, Ulyanovsk, Russia

*Address all correspondence to: V. N. Kovalnogov, E-mail: kvn@ulstu.ru

The results of development and numerical investigation of thermal protection of the surfaces immersed in a disperse working fluid flow in relation to turbomachine blades are presented. The possibility of increasing the accuracy of prediction of the thermal state of blades by obtaining reliable data with the aid of the developed mathematical model and the unique complex of modeling programs is shown. The possibilities and conditions of increasing the efficiency of turbine blades cooling by using damping cavities are investigated.

KEY WORDS: thermal protection, disperse medium, turbine blade, cooling, modeling

1. INTRODUCTION

The movement of disperse flow (of a gas with dispersed solid or liquid particles of condensed phase distributed in it) is implemented in the flowing part of solid-fuel rocket engines, and in gas-turbine and steam-gaseous power plants with complete or partial use of crushed mineral coal, etc. as fuel. For thermal protection of the elements of such constructions immersed in a high-temperature disperse flow of a working medium, internal cooling is widely used by creating a low-temperature curtain near the wall.

The presence of the particles of the condensed working medium in the flow substantially complicates the process of its interaction with the surface of the flow part of power plants. A considerable number of works are devoted to the study of the processes of heat transfer between the surface and disperse flow [for example, (Leontiev et al., 2009; Laptev et al., 2015; Varaksin, 1998; Eaton and Fessler, 1994; Terekhov and Pakhomov, 2004)], but in the majority of cases the surface of straight channels of uniform cross section are considered. Only a few works [for example, (Varaksin, 1998; Zaichik and Alipchenkov, 2005)] are devoted to the study of the effectiveness of a curtain in disperse flow.
The flow part of rocket engines, gas turbines, heat exchangers, and of other thermal and power equipment is characterized by a complex shape. When a disperse flow of a working medium moves in such devices, the conditions for transverse movement of condensed particles in the boundary layer and their inertial loss on certain sites of the surface are created that significantly intensify the heat transfer processes. Thus, when the ratio of the rates of gas and condensed phases in disperse flow is equal to one, the presence of particles in flow practically does not influence the heat transfer intensity in straight pipes of uniform cross section. At the same concentration of particles in the conditions of their inertial loss on a surface, the double intensification of heat transfer was observed in nozzles, fourfold in curvilinear channels, sixfold in pipes with swirled flows (Kovalnogov, 1996).

One of the major factors influencing heat and mass transfer near the wall and distribution of isentropic wall temperature is diffusion and migratory deposition of drops from a turbulent flow on the wall with formation of a liquid film.

2. CHARACTERISTIC FEATURES OF SIMULATION OF DISPERSE FLOW

Disperse flows are encountered in many natural and technical conditions, and almost always they are turbulent. Currently, two-phase turbulent flows represent one of the most rapidly developing branches of mechanics and heat transfer science. This review presents the modern methods of modeling two-phase disperse turbulent flows and statistical models based on the kinetic equations for the probability density function of the velocity and temperature of particles of disperse phase. The calculation of two-phase flow must include the modeling of mass, momentum, and heat transfer for each of the phases, as well as for the interfacial interaction. Basic fundamental difficulties arising in construction of the theory of disperse two-phase turbulent flows are associated with the turbulent nature of the medium motion and with the interaction of particles with one another and with the bounding surfaces. It should be noted, first of all, that to date even the construction of the theory of single-phase turbulent flows is far from being completed, although for describing such flows a number of rather effective models have been suggested and the calculation of many of them do not cause any fundamental difficulties. Despite the fact that the first work on the theory of disperse turbulent flows appeared relatively long ago, the intense developments in the field of the mechanics and heat transfer started only in the last 20 years. The main theoretical problems arising in modeling two-phase disperse turbulent flows compared with single-phase ones relate to the following physical processes: interaction of particles (droplets, bubbles) with turbulent solid-phase eddies (Eaton and Fessler, 1994); interaction of particles with one another as a result of collisions; evolution of particles size range as a result of combustion, phase transitions, coagulation or crushing; influence of turbulent fluctuations on the rate of heterogeneous combustion and phase transitions; interaction of particles with a surface bounding the flow and precipitation; the
inverse effect of particles on the turbulence; dispersion fluctuations and accumulation of particle concentration (Zaichik and Alipchenkov, 2005; Zaichik, 2006).

All the varieties of currently existing mathematical models of disperse flows can be divided into two classes (types). Models of the first class (two-fluid) describe the motion of carrier gas flow and the motion of the multitude of suspended particles. Another type of models (Euler–Lagrange) includes the models in which the motion of a homogeneous gas is considered, while the particles are integrated along their trajectories. The advantages and disadvantages of the two approaches are described in (Zaichik and Alipchenkov, 2005; Zaichik, 2006).

The advantage of the two-fluid models is that numerical models of disperse flows differ little from pure gas models. This affords an opportunity for borrowing numerical methods used for solving the system of equations of a pure gas. The disadvantage of the model of this type is that they carry considerably less information about the motion of a single particle. The disadvantages of the first type of models are the benefit of the second type. The advantages of the Euler–Lagrange models include detailed statistical information on the motion of each particle individually in the known field of velocities and temperatures of the carrier flow. But there are some limitations in the implementation of this type of calculation. On increase in the concentration of the second phase, the degree of the influence on the parameters of the carrier gas particles increases as well as the probability of collision of particles, which leads to the "entanglement" of the trajectories of particles. With a decrease in the size of particles the computational process is complicated, since for correct representation of the motion of particles it is necessary to take into account their interaction with all smaller turbulent eddies in the carrier flow.

3. CHARACTERISTIC FEATURES OF HEAT EXCHANGE AND GAS DYNAMICS IN TURBOMACHINES AND RESERVES OF IMPROVEMENT OF THERMAL PROTECTION

When a gas flows past a turbine blade, a laminar boundary layer is formed on its inlet edge. Under the influence of various factors, vortex disturbances appear in the laminar boundary layer called the Tollmien–Schlichting waves. They can be generated due to the roughness of the surface immersed in flow or be caused by external factors such as acoustic waves or turbulence of the external flow. Moreover, also vibration of the surface immersed in flow can be the cause of vortex disturbances in the laminar boundary layer.

At low Reynolds numbers laminar flow remains stable. When the critical Reynolds number is reached at a certain point, vortex disturbances begin to grow, and the laminar boundary layer loses its stability. Determination of the extent of areas on the blade profile with laminar, transitional, and turbulent flow is important for determining flow patterns and heat transfer in each region that influence the dis-
tribution of local heat transfer coefficients along the blade profile from the gas to the blade surface. In a gas turbine engine with moderate initial temperatures of the gas the Reynolds numbers of the gas turbine blades are typically equal to $10^5$–$10^6$, whereas in high-temperature gas turbine engines they reach the values $9\cdot10^6$. The average, over the perimeter of the profile, value of the heat transfer coefficient on the gas side is 200–1200 W/(m$^2$·deg), and in high-temperature gas turbine engines — up to 5000 W/(m$^2$·deg).

In the boundary layer, the processes of transition not only from the laminar mode to turbulent are possible, but also the inverse processes called relaminarization. Inverse transitions occur only at sufficiently large negative pressure gradients, in the case of longitudinal nonisothermicity etc. Moreover, a significant effect on the processes of heat exchange is exerted by the boundary layer separation from the airfoil surface. This phenomenon is currently poorly understood, therefore the simulation is carried out approximately. Separation zones appear under the influence of a positive pressure gradient, shocks, blowing on the surface, etc.

In transition the temperature factor should be taken into account. It is shown in (Zaichik, 2006) that when the temperature difference between the wall and the air flow is $\Delta T = 100^\circ C$, the Reynolds number of the start of transition decreased twice as compared to the isothermal boundary layer. For the process of flow past a plate with the surface temperature less than the air flow temperature, the velocity profile over the boundary layer thickness is convex, without the inflection point.

Consequently, the cooling of the surface being streamlined by hotter air promotes flow stabilization like the negative pressure gradient, whereas the heating of the surface causes a destabilizing effect like the positive pressure gradient. At the wall temperature less than temperature of the air flow, the critical Reynolds number is 1.5 times higher in comparison with the process in which these temperatures are equal (Shvets and Dyban, 1974).

Because of the complex structure of the boundary layer on the blade surface, it is advisable to use the concept of effective viscosity in the boundary layer $\nu_{\text{ef}}$. In a laminar boundary layer $\nu_{\text{ef}} = \nu$, i.e., is equal to the ordinary kinematic viscosity coefficient that depends on its temperature and pressure; in a turbulent boundary layer, as outlined above, $\nu_{\text{ef}} = \nu + \nu_T$; in a transitional boundary layer $\nu_{\text{ef}} = \nu + \gamma \nu_T$, where $\nu_T$ is turbulent viscosity and $\gamma$ is the intermittency factor in the transitional zone. Thus, to calculate the local heat transfer on the profiles of turbine blades by solving differential equations of boundary layer with the use of the mixing path length model, it is necessary to determine the coordinates of the beginning and end of the transition of the laminar boundary layer into a turbulent one, and then to find the length of the sections of the profile with different flow regimes, to determine the intermittency factors in the transitional zone, and, after the solution of the system of differential equations with the use of the dependencies for the effective viscosity, to find the distribution of velocities and temperatures in the boundary layer, the local
values of the specific heat flux on the surface immersed in flow, and the corresponding heat transfer coefficients.

With the use of different types of penetrating cooling with blowing out of a coolant to the surface it is necessary to consider the effect of blowing on the boundary layer characteristics. Blowing can be organized in various ways, including penetrating porous cooling.

The methods for calculation of boundary layers on permeable surfaces with continuous distribution of injection have been developed rather adequately, when the problem can be solved within the framework of the approximate boundary layer theory. Such problems are solved in the framework of the Navier–Stokes equations or of their turbulent analog — the Reynolds equations. Moreover, heat transfer in the blades is affected by additional factors attributable to the rotor rotation: the centrifugal and Coriolis forces.

To carry out an analysis of hydrodynamic and thermal processes in ducts of complex shape, two-dimensional and even three-dimensional equations of motion and energy should be considered, since heat can be transferred in such channels by means of secondary flows in the transverse section of the channel.

The characteristic feature of hydrodynamic and thermal processes in channels of complex shape is that the cooler flow rate in the transverse section of the channel is substantially nonuniform, which leads to significant changes in the shear stress, heat flux, and surface temperature over the whole perimeter of the transverse section of the channel.

The temperature field in the cooler flow and on the surface is determined by the thermal interaction of flow with the surface, therefore it is necessary to analyze a conjugate problem.

To determine friction and the cooler flow rate distribution, an approximate engineering computational method can be used.

Numerous experiments have shown that for a developed turbulent stabilized flow of liquid in channels of various shapes the character of the dependence of the resistance coefficient on the Reynolds number is the same as for flow in a cylindrical channel.

4. MATHEMATICAL MODEL

The unique results of investigation of the temperature stratification will be included into the SolidWorks software package. This work is being carried out at the Ulyanovsk State Technical University on the basis of the TurboWorks package that was developed herein (Koval'nogov and Koval'nogov, 2003). This will improve the accuracy of prediction of the thermal state of turbine blades, as well as the efficiency of cooling systems.

For complete and high-quality calculation of a three-dimensional dynamic temperature profile of a blade in nonlinear formulation with account for the temperature de-
pendence of the thermal properties of blade material use is made of numerical methods. It is necessary to solve the spatial nonlinear problem of the thermal conductivity of the blade.

The system of equations describing the stationary processes of motion and heat transfer of turbulent flow in a pipe has the form (Kovalnogov and Fedorov, 2015)

- differential equation of heat transfer

\[
\alpha = - \frac{\lambda}{T_{w_0} - T_w} \left( \frac{\partial T}{\partial y} \right)_{y=0};
\]

- differential equation of the boundary layer energy

\[
\rho c_p \left( u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} \right) = \frac{1}{r} \frac{\partial}{\partial y} \left[ r (\lambda + \lambda_T) \frac{\partial T}{\partial y} \right];
\]

- differential equation of the boundary layer motion

\[
\rho \left( u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = \rho_u u_{\infty} \frac{du_{\infty}}{dx} + \frac{1}{r} \frac{\partial}{\partial y} \left[ r (\mu + \mu_T) \frac{\partial u}{\partial y} \right];
\]

- differential equation of continuity

\[
\frac{\partial (\rho u)}{\partial x} + \frac{1}{r} \frac{\partial (\rho v)}{\partial y} = 0;
\]

- equation of state

\[
\rho = \frac{p}{RT},
\]

where \( \alpha \) is the heat transfer coefficient, \( T \) is the temperature, \( u \) and \( v \) are the longitudinal (along the axial coordinate \( x \)) and transverse (along the \( y \) coordinate) components of the flow velocity, respectively; \( r \) is the radius of a point analyzed; \( \rho, c_p, \lambda, \) and \( \mu \) are the density, specific isobaric heat capacity, thermal conductivity, and dynamic viscosity of flow, respectively; \( \lambda_T \) and \( \mu_T \) are the coefficients of turbulent heat and momentum transfer, respectively; \( p \) is the pressure, and \( R \) is the gas constant. The subscript \( \infty \) refers to the parameters on the axis of the tube and the subscript \( w \) describes those on the surface in the flow part.

The boundary conditions are

\[
x = 0 : u = u_0; \quad T = T_0;
\]

\[
y = 0 : u = 0; \quad v = 0; \quad T = T_w;
\]

\[
y = R_1 : u = u_{\infty}; \quad T = T_0.
\]
Here $R_1$ is the radius of the flow part of the pipe; the subscript 0 refers to the parameters at the pipe inlet.

The velocity $u_\infty$ in each section of the pipe is determined by the relation

$$u_\infty = \frac{u_0 \left( \frac{\rho_0}{\rho_\infty} \right)}{1 - 2 \frac{\delta^*}{R_1}},$$

where the displacement thickness $\delta^*$ is expressed by the formula

$$\delta^* = \frac{R_1}{\int_0^1 \left( 1 - \frac{y}{r} \right) \left( 1 - \frac{\rho u}{\rho_\infty u_\infty} \right) dy}.$$ (8)

Because of the uncertainty of the quantities $\lambda_T$ and $\mu_T$, the mathematical formulation of the problem is not closed. For its closing, we will avail ourselves of the Prandtl algebraic turbulence model. In accordance with the Prandtl mixing-length model the coefficient of turbulent transfer of momentum $\mu_T$ is expressed by the relation

$$\mu_T = \rho l^2 \frac{\partial u}{\partial y},$$

where the mixing length $l$ can be calculated from the expression:

$$l = \alpha y \left[ 1 - \exp \left( -\rho u^* y / (2\rho) \right) \right].$$

where $u^*$ is the dynamic velocity at the point considered and $\alpha$ is the coefficient characterizing the intensity of the turbulent transport of momentum.

The dynamic velocity is determined as

$$v^* = \sqrt{\tau / \rho},$$

where $\tau$ is the shear stress at the given point:

$$\tau = (\mu + \mu_T) \frac{\partial u}{\partial y}.$$ (12)

For standard conditions, the coefficient $\alpha$ is considered to be a constant value equal to 0.4. However, as shown in (Koval'nogov, 1996; Koval'nogov and Fedorov, 2011), in flows under external actions, this factor can undergo significant changes and differ from this value. Based on the experimental data (Fig. 1), the form of the coupling of the coefficient $\alpha$ with the parameter of turbulent pressure fluctuations of air flow in a perforated pipe with damping cavities $H^*$ has been established:

$$\frac{\alpha}{\alpha_0} = A \cdot \theta \left( C H^* + D \right) + B,$$ (13)

where $A$, $B$, $C$, and $D$ are the empirical coefficients of closure. The empirical coefficients were determined by linking the results of calculation by a numerical method with experimental data by the method of least squares.
It is assumed that the mechanisms of substances and momentum transfer are identical. The turbulent heat transfer coefficient is determined as

$$\lambda_T = \frac{\mu_T c_p}{\text{Pr}_T} \approx \frac{\mu_T c_p}{0.9}. \quad (14)$$

Heat transfer calculations were performed by the empirical similarity equation:

$$\text{Nu}_{xw} = 0.029\text{Re}_{xw}^{0.8}\text{Pr}_w^{0.4}\left(\frac{T_w}{T_r}\right)^{0.39} \times \left(1 + \frac{k - 1}{2} M^2\right)^{0.11}. \quad (15)$$

Here, the Nusselt number is defined by the expression

$$\text{Nu}_{x0} = \frac{\text{Nu}_{xw}}{\text{Pr}_w^{0.4}\left(\frac{T_w}{T_r}\right)^{0.39}} \times \frac{1}{\left(1 + \frac{k - 1}{2} M^2\right)^{0.11}}. \quad (16)$$

The investigations have shown that the error of numerical calculation of the heat transfer coefficients caused by the inaccurate determination of the temperature recovery coefficient increases with the Mach number M and decreases with the temperature factor. The effectiveness of film cooling of the surface is found from the formula

$$\theta = \left(T_r - T_{adw}\right) / \left(T_r - T_{w3}\right). \quad (17)$$
where $T_r$ is the temperature of the heat-insulated surface in the absence of the curtain (temperature of flow "recovery"); $T_{adv}$ is the same in the presence of the curtain; $T_{w0}$ is the surface temperature in the initial section of the protected surface area (the initial cross section of the protected area coincides with the end section of the permeable area where blowing is carried out and a low-temperature curtain is formed).

The adequacy of the model of turbulent disperse boundary layer was verified by comparing the calculations of the heat transfer coefficients of disperse flow in nozzles with experimental data. The results of comparison (Kovalnogov and Fedorov, 2011) show that the proposed technique as a whole adequately reflects the characteristic features of transfer processes in a dispersed boundary layer.

**5. TESTING AND ANALYSIS OF THE ADEQUACY OF THE PROCEDURE**

The characteristic features of the numerical integration of the system of boundary layer equations in investigating the effectiveness of a curtain stem from the fact that transverse gradients of velocity and temperature differ significantly in the wall region. The large gradients of longitudinal velocity near the wall on the heat-insulated portion require the use of a fine grid to reduce the error of approximation of differential operators by difference ones, whereas the low temperature gradients at the same time require the use of a coarse grid to reduce the rounding errors in calculation of transverse coordinate derivatives. There are the same contradictory requirements to mesh sizes in relation to two neighboring areas: the permeable section where the curtain is formed and the adiabatic section.

The error of the numerical method is attributed to the replacement of the initial equations that describe the adopted model of the physical phenomenon by other approximating equations allowing one to construct a computational algorithm, as well as to the approximate the nature of the methods of solving these equations. Usually, numerical methods are constructed with the use of a certain parameter; when this parameter approaches a definite limit, the error of the algorithm tends to zero. The computational algorithm is also verified by a system of tests. The problem of containing specific difficulties of the given class of problems with known exact solutions serves as a test.

The error of rounding off owes its origin to the fact that any calculation on a computer or manual calculations are performed with a limited number of significant digits. When performing one arithmetical operation with numbers, the rounding error lies within the unit of saved junior category. In this way the personal computer operates with numbers usually containing 10–12 bits, therefore the error of single rounding is negligibly small in comparison with the nonremovable error. When calculating on a PC, billions of operations can be executed, but if there are no systematic reasons for accumulation of rounding errors, they do not increase substantially, since in various operations the errors will have different signs and will cancel each other. Nevertheless, if the numerical method is such that there arise systematic reasons for the ac-
cumulation of rounding errors, the total error increases very rapidly to catastrophic proportions and the obtaining of reliable results becomes impossible.

In order to determine the accuracy of the numerical solution the error of calculation of the heat transfer coefficients of the working fluid and coolant has been determined. In simulating the thermal state of the blade with the aid of the developed program-information complex, four components of the error of numerical solution are singled out: the error in determining the coefficient of heat transfer from the side of the working fluid on the back and the trough of the blade; the error in determining the coefficient of heat transfer from the coolant; the error in determining the coefficient of heat transfer from the working fluid for the frontal edge of the blade, and the error of calculation in determining the heat transfer coefficient from the working fluid for the trailing edge of the blade.

The procedure of finding errors in the minimum and maximum temperatures of the blade consists of carrying out of the initial calculation (calculation of the boundary conditions and of the blade temperature field), which is accepted as the true solution, provided there is no error of numerical solution. After that, the values of the heat-transfer coefficients for one of the characteristic surfaces of a blade (the surface of the cooling way, the trough, the back, and the leading or trailing edge) deviate by a value of the error \( \pm P \) [%].

Next, the adequacy of the developed mathematical model was checked by comparing the data on the distribution of temperature and heat transfer coefficients over the surface of the trough and back with the experimental data obtained at the Kazan Aviation Institute (Kovalnogov, 1996). Moreover, we compared the results of the calculation of the heat transfer coefficients on the back and trough. The calculations were performed in conjugate formulation. Based on the analysis, it has been found that the model has a satisfactory accuracy. The characteristic distribution of temperature along the contour of the back of the blade was calculated before the establishment of temperatures with an error of numerical integration of 1%. Based on the comparison of temperatures on the blade back with experimental data a satisfactory accuracy of up to 21 K is achieved, which indicates the adequacy of the developed mathematical model and the possibility of determining the thermal state of turbomachinery blades in CAD.

A comparison of the results of heat transfer calculation is given in (Koval'nogov and Fedorov, 2011) for a uniform air flow at subsonic and supersonic velocities and for two-phase flow. The proposed model and method of calculation are adequate to the characteristic features of heat transfer for disperse and homogeneous flows. The discrepancy between the calculated results and experimental data does not exceed 25% for the Nusselt number and 3% for the temperature recovery coefficient.

6. RESULTS OF INVESTIGATIONS

The numerical investigation was made in relation to the blade of a turbomachine streamlined by high-speed turbulent disperse flow with air as the carrier medium with
account for the temperature dependence of the thermophysical properties of the carrier medium. The results of the calculations do not exceed the error of determining the temperature in the stage of designing. This is confirmed by the values of the maximum temperature depending on quantity of calculating elements given in Fig. 2.

Figure 2 shows that satisfactory accuracy of calculation is achieved when the number of calculating elements is over 80,000. Therefore computations were made for the blade consisting of 80,000 elements. The calculation results are presented in the form of thermal graphs. Figures 3 and 4 show that the blade edges are the most thermally loaded.

At the same time due to convective cooling the temperature of the back and trough is lower than the temperature of the edges by more than 100 K. The minimum temperature in the middle section of blade is 1248 K, with 1255 K at the end face. The nonuniformity of the warming of the blade is a significant drawback that reduces the term of blade operation.

The next series of calculations was carried out in relation to cooling with application of the perforated surfaces of the trough and back with damping cavities (Kovalnogov and Kovalnogov, 2004; Kovalnogov et al., 2015). Figure 5 shows that through the use of perforations on the surface, as well as of the damping cavities, the efficiency of the curtain increases due to the change in the area of the perforations \( f \), without increase in the coolant flow rate.

That the flow could not to be disturbed, the diameter of the perforations must be small. The curtain is stabilized by decreasing turbulent pulsations and also turbulent

![FIG. 2: Maximum temperature of the blade on different grids with account for the dependence of the thermal properties of the material on the blade temperature](image-url)
transfer in the area near the wall, which is reached by applying the perforations connected with damping cavities.

Figures 6 and 7 show the effect of the number of holes in the damping cavities on the efficiency of film cooling along the trough and the back of the blade. An increase in the number of holes $n$ is accompanied by proportional increase in the relative area. The maximum efficiency of film cooling is achieved at $n = 2$. The same applies to the intensity of heat transfer that has a maximum value at $n = 2$. A further increase in the number of holes leads to a decrease in the efficiency of film cooling. Because of the high temperature gradient the heat stresses can exceed the blade material strength. The expanded channels of air cooling lead to a situation that blades operate safely below their material safety temperature limit. The technologies of casting allow one to make thin output edges as converging channels for decreasing the weight and ensuring the best aerodynamic properties. But this area of the blade is subjected to large

FIG. 3: Thermal picture of projected blades, depending on the duration of heating (cross section): a) 20 s; b) 120

International Journal of Energy for a Clean Environment
FIG. 4: Thermal picture of projected blades, depending on the duration of heating (3D): a) 20 s; b) 120 s

aerodynamic, thermal, and structural stresses, therefore the local information on heat transfer is of crucial importance for prevention, modification, and optimization of surface cooling.

7. CONCLUSIONS

• Based on the study carried out, the possibility of using the developed program-information complex as an instrument for high-precision analysis of the thermal state of turbomachinery blades is shown. Numerical studies of different
methods of thermal protection, including the perforated surfaces of the trough and back of the blades with damping cavities during convective film cooling have been carried out.

FIG. 5: Effect of the relative area of perforations on the change in the efficiency of film cooling of the blade along the trough (a) and back (b) on a perforated surface with damping cavities: $\bigodot, f = 0$; ■, 0.001; $\triangle$, 0.0015; ◊, 0.002
FIG. 6: Effect of the number of holes in the damping cavities of a trough on the efficiency of film cooling on a perforated surface (a) and on the heat transfer coefficient in the curtain area (b): ○, n = 0; □, 1; △, 2; ▲, 3; ●, 4; ■, 5

- It has been found that the use of perforated cavities can increase the maximum efficiency of film cooling by about 1.8 times, with the best results being obtained when the number of holes on each damping cavity is \( n = 2 \).
ACKNOWLEDGMENT

The study was supported by the Russian Foundation for Basic Research, research project No. 16-31-06114 mol_a dk.

FIG. 7: Effect of the number of holes in the damping cavities of the back on the efficiency of film cooling on a perforated surface (a) and on the heat transfer coefficient in the curtain area (b): ○, n = 0; □, 1; △, 2; ▲, 3; ●, 4; ■, 5
REFERENCES


