THE INFLUENCE OF THE SHAPE OF THE COOLING CHANNEL AND PRESSURE GRADIENT ON THE EFFICIENCY OF FILM COOLING OF GTE BLADES

S. V. Belov, G. B. Zhestkov, D. V. Muravsky , S. V. Kharkovsky, E. V. Shcherbakova.
Central Institute of Aviation Motors (CIAM), Moscow, Russia

Keywords: *blade, film cooling, convective cooling, fan-shaped holes*

Abstract

The purpose of this work is to improve numerical methods for studying the thermal state of gas turbine engine (GTE) blades with film cooling of the outer surface of a blade blown by a hot gas and convective cooling of its internal surface.

1 Formulation of the problem

At the present stage of development of aircraft technology, the efficiency and reliability of the engine is largely determined by the thermal state of one of the key units - a high-temperature turbine and its elements, primarily nozzle and working blades. The permissible level of the thermal state of the blades is ensured by the use of effective systems of convective-film cooling and by the intensification of heat exchange inside the blades by means of placing on the walls of the internal cooling channels the blades of various heat exchange intensifiers such as dowel matrices, wells, annular swirlers.

Carrying out a full three-dimensional calculation of the thermal state of the blade with a developed system of convective-film cooling in conjunction with the gas dynamic calculation of the external flow around the blade and the calculation of the flow in all internal cooling channels is a very complex and time-consuming task even with modern computer equipment. In addition to the enormous computational power required to calculate such a problem, there are fundamental limitations connected with the imperfection of the empirical models of turbulence used in gas dynamically complex flow areas, such as circulation zones, regions with large pressure gradients. Therefore, at the present time the papers dealing with individual aspects of internal [1-3] and external [4-5] flows are most widely used.

In the previous works of the authors [4-5], the cooling of the entrance edge of the nozzle and working blades of the turbojet was investigated. The comparison with its own experiments have shown that quantitative agreement between calculation and experiment is necessary to solve conjugate heat transfer problem.

This work is a continuation of previous studies, extended to the rest of the surface of the GTE blades. We compare of the cooling performance at blowing the cooling air from the cylindrical and fan-shaped holes. Further, we investigate the influence of a pressure gradient in the external flow generated by the curvature of the channel walls on efficiency of a film cooling. In addition, the calculation of a linear cascade is carried out in the presence of several rows of cooling air blowing.

1.1 Initial equations, calculation method and similarity criteria

During the calculations, we used the software complex CFX, based on the solution of the complete three-dimensional Reynolds-averaged Navier-Stokes equations. The basic dimensionless parameters determining the flow of a viscous compressible gas in the computational domain are the complex of dimensionless geometric parameters characterizing the calculated region and well-known similarity parameters - the Reynolds and Mach numbers, the ordinary and turbulent Prandtl number, the temperature factor $T_w=T_{w0}/T_0$ (T_{w0} and T_0 are the characteristic wall and gas flow temperatures). In the presence of cooling air blowing, the new dimensionless parameters are added to the already listed dimensionless parameters, characterizing the intensity and geometry of the blowing. The most important geometric parameters are the dimensionless diameter of the holes D, the relative step between the holes $L=L_b/D$ (L_b is the dimensionless hole spacing), and the blowing angle α between the cooled wall and the blowing channel. The intensity of blowing is characterized by the parameter $M=(\rho\omega)_c/(\rho\omega)_h$ (ρ and ω is the characteristic density and velocity , the indices "c" and "h" refer to the blowing parameters and the main gas flow, respectively), and the temperature characteristics - dimensionless blowing temperature $T_c=T_{c0}/T_0$ (for simplicity we assume that the chemical composition of the main and blown gas is the same).

When considering the conjugate heat exchange problem between the wall and the gas flow, new dimensionless parameters are added that characterize the geometry of the wall and the difference in the thermal diffusivity $A=(\lambda/(\rho_0 C_p)$ between the wall material and the gas. In this expression, λ is the thermal conductivity and C_p - the specific heat at constant pressure. The most important are the dimensionless wall thickness h and the ratio of the thermal diffusivity coefficients $A^0=A_c/A_w$ .We note that if the coefficient of thermal diffusivity of the wall is practically constant and depends mainly on the wall material, the coefficient of thermal diffusivity of air can vary over a wide range due to a change in the characteristic density of the cooling air. The increase in the density of air should increase the intensity of film cooling

To close the Reynolds equations, a two-parameter SST Wilcox turbulence model is used, based on solving the equations for k and ω. This model is supplemented by corrections that allow one to take into account the compressibility of the flow and the curvature of the streamlines. The transition from laminar flow to turbulent flow was described using a transit turbulence model based on solving the algebraic expression (Specified Intermittency), [6-8].

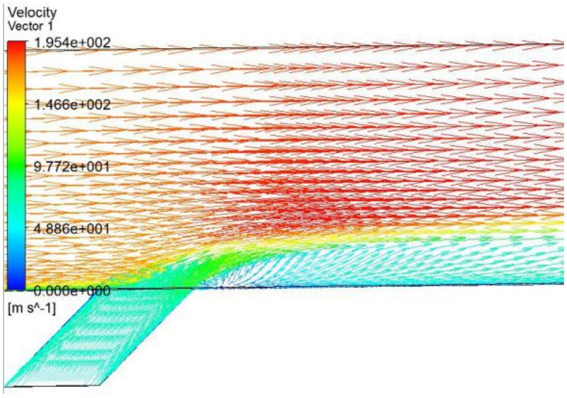
2 Results of calculating the conjugate heat transfer problem

2.1 Blowing the cooler on a plane surface from cooling channels of various shapes.

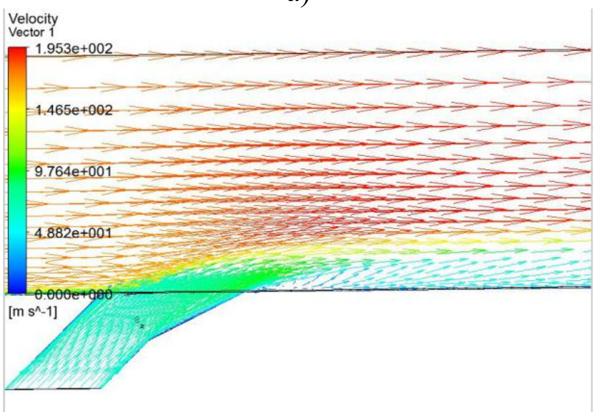
The calculation domain for numerical simulation of gas flow with blowing of cooling gas through a series of perforation holes consists of a channel of rectangular shape with a height h and a width H through which hot air flows and walls of thickness h1 with a series of cooling channels. To save the calculation time, the calculation domain is bounded by flow symmetry planes running along the direction of the main flow perpendicular to the cooled channel wall through the plane of symmetry of the perforation channel and at an equal distance between the symmetry planes of adjacent perforation channels. In the inlet section of the channel, the total pressure, the total temperature and the inflow angle were set ($P_0^* = 1.1*10^6$ n /m^2, $T_0^* = 700$ K, $\alpha = 0$), at the outlet section - static pressure ($P = 10^6$ n /m^2), in the initial section of the blowing channel - mass flow , temperature (G = 0-0.06 g / s, T = 300 K) and the cooling air blowing angle - parallel to the channel walls in the initial section. We considered a cylindrical perforation channel with a diameter D = 1.4 mm, located at an angle of 45 ° to the cooled channel wall, and a fan-shaped holes of the same initial diameter, with an exit angle of 27 °. Since in the real gas turbine blades the common cooling air supply channel is located perpendicular to the design area and the heating of air when passing such a small distance is insignificant, it was considered that the wall temperature from the cooling air side is constant (in calculations $T_w = 550$K). The basic dimensionless (relative to D) geometric parameters of the computational domain are:

- distance between perforation holes H = 2.84;
- channel height h = 3.55;
- length of the initial section L1 = 8,
- length of output section L2 = 22,
- dimensionless wall thickness h1 = 1.42

Referring to Fig. 1 shows the directions of the velocity vector in the plane of symmetry of the flow that runs along the direction of the main flow perpendicular to the cooled channel wall through the plane of symmetry of the perforation channel. There are no significant features of the flow within the channels of perforation.



a)



b)

Fig. 1 Directions of the velocity vector in the plane of symmetry in the region of the air blowing point: a) from the cylindrical channel, b) from the channel with the fan-shaped holes

There are small circulation zones on the cooled channel wall behind the blowing site, and in the case of blowing out of the fan-shaped holes this zone is somewhat smaller. The loss of total pressure in both cases is the same

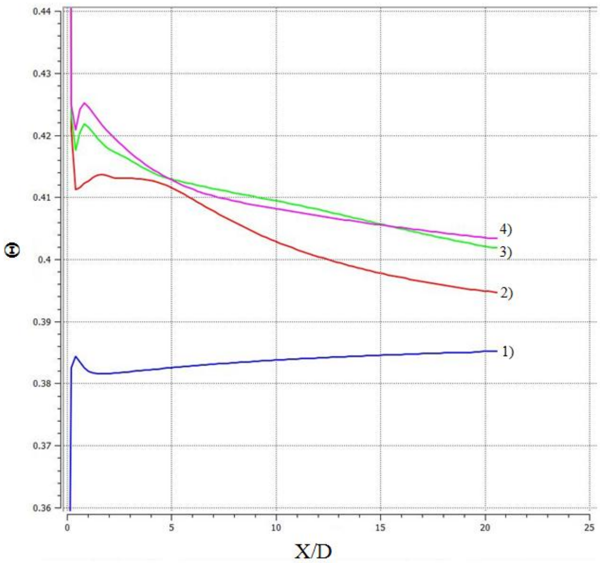


Fig. 2 Dependence of the dimensionless temperature Θ on the distance X / D from the blowing point from the cylindrical hole:
1) M = 0; 2) M = 0.25; 3) M = 0.5; 4) M = 0.75

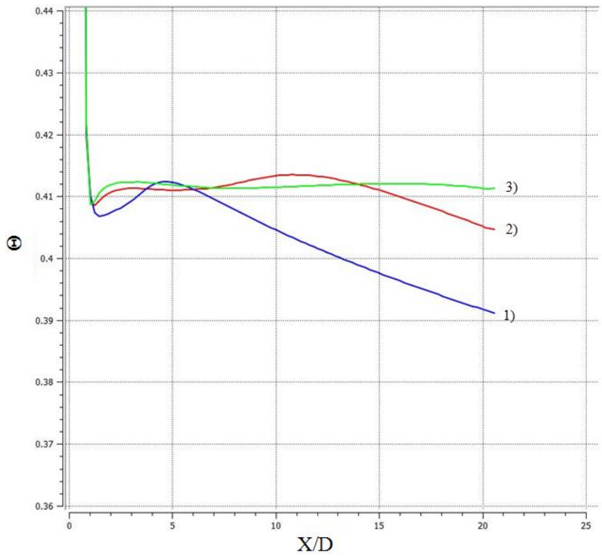


Fig. 3 Dependence of the dimensionless temperature Θ on the distance X / D from the blowing point from the fan-shaped hole:
1) M = 0.25; 2) M = 0.5; 3) M = 0.75

Figures 2 and 3 show the dependences of the dimensionless temperature $\Theta = (T_h^0 - T) / (T_h^0 - T_c^0)$ from the distance X / D from the blowing point along the line located in the plane of symmetry of the perforation channel at different blowing rates from the cylindrical channel and the channel with the fan-shaped holes .

The results of calculations (Fig. 1) follows that the primary cooling tunnel wall in contact with hot air is achieved at the expense of the thermal conductivity of-for cooling other surface.. The influence of the protective film is insignificant and increases somewhat when blowing out of the cylindrical channel with increasing intensity to M = 0.5.

Further increase in the intensity of blowing is ineffective. Blowing out from the channel with a fan-shaped holes is less effective at a distance of less than 10 calibers from the blowing site than from a cylindrical channel.

In cooling systems of gas turbine blades, the pressure and the cooling temperature of the cooling air are usually known. Therefore, comparative calculations were carried out at the same pressure and cooling temperature of the cooling air in the initial section of the blowing. Under these boundary conditions, the cooling efficiency is everywhere better when blowing cooling air out of the fan-shade hole, because the cooling air flow through the fan-shade hole is approximately 13% greater than the air flow through the cylindrical hole. This ratio is valid throughout the considered range of variation in the parameters of air injection. For reference, we note that the ratio of the output areas of the cylinder and fan-shaped holes is 1.56 and is not proportional to the relative change in gas flow. Apparently, it is possible to estimate the real change in gas consumption with such a significant change in the geometry of the supply channel only from the data of a three-dimensional calculation or experiment. When performing one-dimensional calculations, you need to introduce significant correction factors.

To evaluate the influence of the thermal diffusivity of air, $A = (\lambda / (\rho C_p)$, on the heat transfer characteristics in the calculation of the conjugate problem, an additional series of calculations was carried out under the following boundary conditions: the inlet section of the channel is given the pressure, the deceleration temperature and the inflow angle ($P_0* = 1.1 * 10^6$ Pa, $T_0 = 700$ K, $\alpha = 0$), in the output section - static pressure ($P = 10^6$ Pa), in the initial section of the channel blowing-out flow, temperature (G = 0-0.6 g / s, T = 300 K) and the blowing angle of the cooling air is parallel to the walls of the channel in the initial section. It allows, by increasing the characteristic air density by a factor of 10, in comparison with a series of previous calculations, to increase the dimensionless parameter-the ratio of the thermal diffusivity coefficients $A^0 = A_h/A_w$. It was possible to save the similarity in the Reynolds number by reducing the calculated area by a factor of 10, but, since in this paper we use the parameter values characteristic of real turbine operation modes, the influence of the change Reynolds number will be analyzed further. We note that in the above calculations the Mach number at the entrance to the channel was M = 0.4, and the Reynolds number, determined from the diameter of the hole, $Re = 4*10^3$ and $Re = 4* 10^4$, respectively.

Dependences of the dimensionless temperature Θ on the distances from the blowing point from the cylindrical hole are shown in Fig. 4. It is seen that the influence of the dam wall and heat shroud on its cooling are comparable to each other. In the absence of blowing, the wall is cooled by thermal conductivity to $\Theta = 0.3$, and in the presence of blowing and thermal conductivity - up to $\Theta = 0.5$ using a lower relative air flow than in the previous case. Similar results for blowing air out of a channel with a fan-shaped holes are shown in Fig. 5. When blowing out of both channels under consideration, the cooling intensity generally increases with increasing cooling air flow. At a distance X / D <5, the maximum cooling efficiency is achieved in both cases under consideration with a minimum considered cooling air flow rate and is comparable with each other. A comparison of the cooling efficiency for blowing out from a cylindrical channel and a fan-shaped holes showed that at X / D <5 the cooling efficiency is better for a cylindrical channel, and for X / D> 5 - for a channel with a fan-shaped holes. With a distance of X / D> 10 from the blowing point, the cooling efficiency of the wall when blowing air out of the channel with the fan-shaped holes is 10-15% higher than the similar values for the case of blowing from a cylindrical channel.

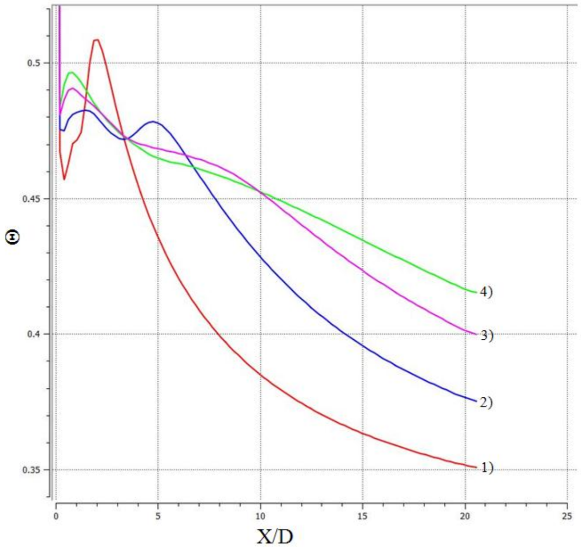


Fig. 4 Dependence of the dimensionless temperature Θ on the distance X / D from the blowing point from the cylindrical hole at P_0 = 1.1*10^6 Pa: 1) M = 0.12; 2) M = 0.25;
3) M = 0.37; 4) M = 0.5.

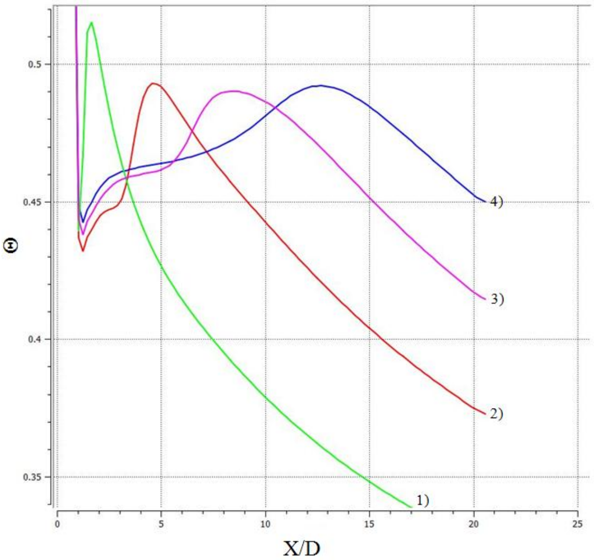


Fig. 5 Dependence of the dimensionless temperature Θ on the distance X / D from the blowing point from the hole with the fan-shaped holes at P_0 = 1.1*10^6 : 1) M = 0.12;
2) M = 0.25; 3) M = 0.37; 4) M = 0.5.

2.2 Blowing the cooler on a curved surface

The calculation domain for numerical simulation of gas flow with blowing of cooling gas through a series of perforation holes consisted of a channel of rectangular shape with a height h and a width H through which hot air flowed and walls of thickness h1 with a series of cooling channels. To save the calculation time, the calculation domain was limited by the flow symmetry planes passing along the direction of the main current perpendicular to the cooled channel wall through the plane of symmetry of the perforation channel and at an equal distance between the planes of symmetry of the adjacent perforation channels. In the inlet section of the channel, the total pressure, the total temperature and the inflow angle are given (P_0* = 1.06*10^6 Pa, T_0* = 1200 K, β = 0), in the output section - static pressure (P = 10^6 Pa), in the initial section of the channel blowing - mass flow, temperature (G = 0-0.2 g/s, T = 300 K) and the cooling air blowing angle - parallel to the walls of the cooling channel. On the outer surface of the air channel, conditions for non-flow and adiabatic flow were set, and the outer surface of the wall was considered to be cooled, and boundary conditions of the third kind - the cooler temperature and the heat transfer coefficient (T = 300K, α = 2000 W*m^2/K) were specified. A cylindrical perforation channel D = 1.4 mm, located at an angle of 45 ° to the cooled wall of the channel. Immediately behind the blowing place was located a curvilinear section with a turning radius R and a turning angle of 45 °, followed by a straight section. The basic dimensionless (relative to D) the geometric parameters of the computational domain:

- distance between perforation holes H = 2.84;
- channel height h = 3.55;
- length of the initial section L1 = 8;
- length of the output section L2 = 22;
- radius of curvature of the concave wall R 14.2 and 7.1;
- radius of curvature of the convex wall R 14.2 and 7.1.

Figs. 6 and 7 show the temperature distributions T in the plane of symmetry of the perforation channel for different curvature of the channel and for the maximum of the considered blowing intensities. In all the cases considered, the circulation zones formed behind the blowing site. The largest zone corresponds to a concave channel, where air is blown into the region of reduced flow velocities, and the smallest - to a convex channel with air blowing into the region of increased flow velocities. This trend is valid for all considered blowing intensities. After the flow is attached to the channel wall, dynamic boundary layers are formed. The thinnest boundary layer is observed in a convex channel, thicker in a rectilinear channel, and in the concave channel the boundary layer is the thickest and even slightly deformed core of the flow. The thermal boundary layers behave in a similar way. It can be clearly seen that when blowing out of the concave channel a portion of the cooling air enters the core of the flow, causing it to cool.

For a quantitative comparison of the results of calculations, it should immediately be noted that, according to the Reynolds analogy, the heat transfer coefficient on the convex wall in the absence of blowing is slightly larger than on the concave wall. On the concave wall there is also less contact surface with hot air and a more cooled surface, on the rectilinear wall they are equal, and on the convex wall the heated surface exceeds the cooled surface. Therefore, if there is no blowing out of the cooler, the concave wall is better cooled and worse - convex. Blowing out the cooler drastically changes the situation. With the channel geometry under consideration, the most effective is the cooling of the convex wall, close in efficiency - a straight wall and the least effective - a concave wall. At a distance Lx / D = 10 on a concave wall, there is no film cooling at all.

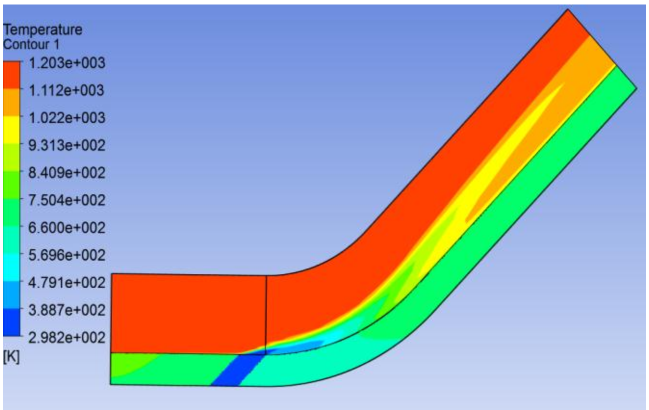


Fig. 6 Temperature distributions in the symmetry plane of the perforation channel with a blowing intensity M = 0.46 is a concave channel.

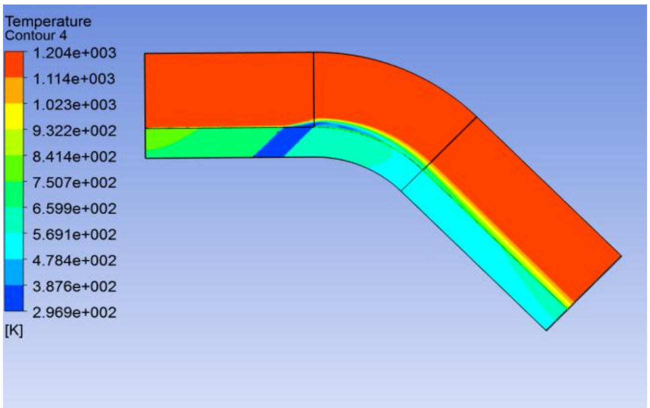
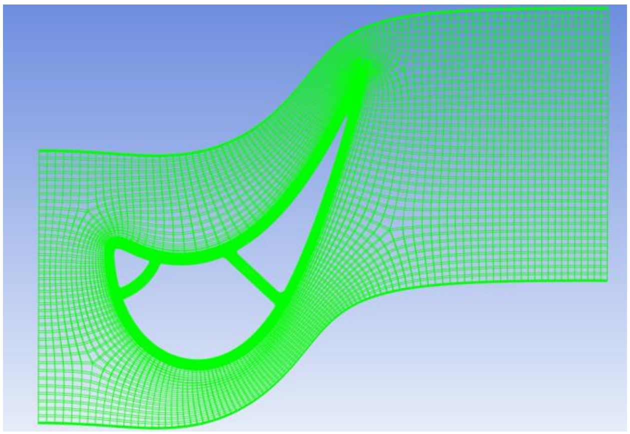


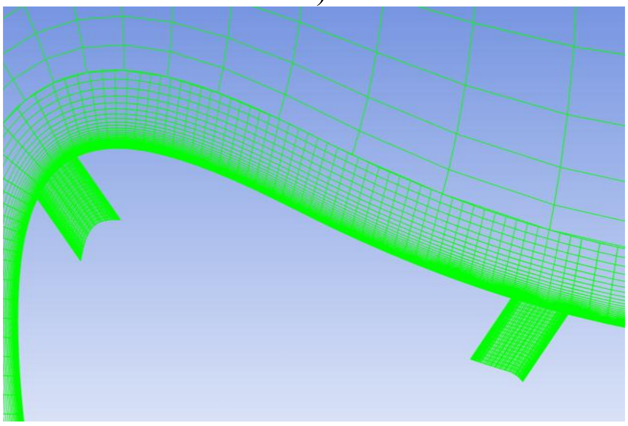
Fig. 7, the temperature distribution in the channel at the plane of symmetry perforation blowing intensity M = 0.46 - convex channel.

2.3 Linear cascade with blowing cooler

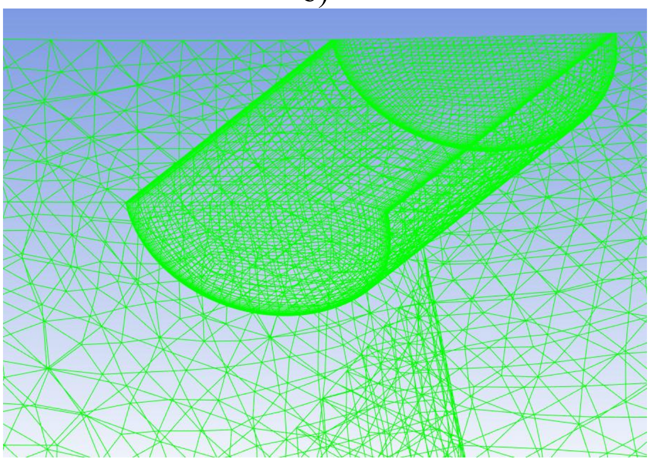
As an example, a linear cascade with six rows of cooling apertures is calculated. Two rows of cooling holes are located on the concave and convex surfaces of the blade, one row at the critical point and one at the output edge. A composite computational grid was used. In a solid, a tetrahedral grid was constructed using ANSYS Meshing. In the area of potential flow in the intervane channel, a regular calculation grid was constructed using ANSYS Turbogrid, and in the boundary layer and in the blowing channels there were regular calculated grids constructed in the ICEM CFD. Computational grid are shown in Fig.8. The results of the calculations are shown in Fig.9.



a)



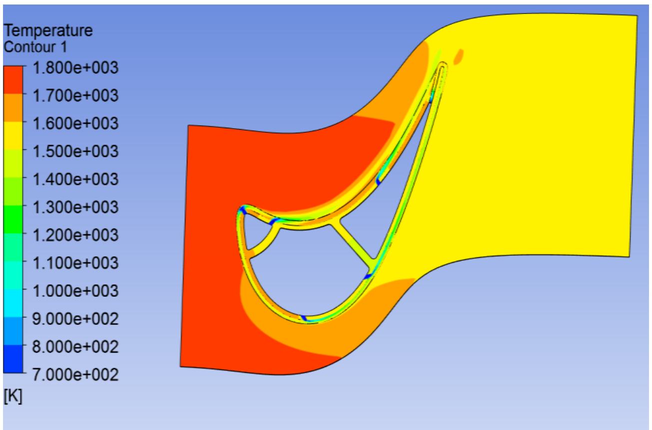
b)



c)

Fig. 8 Computational grid: a) general; b) boundary layer; c) hole and solid.

The main qualitative result of the calculations is the behavior of the boundary layers on the back and trough of the blade repeating the behavior of the boundary layers investigated in the previous chapter on model problems. In this example, the blade back is cooled much better than the trough due to much more efficient film cooling.



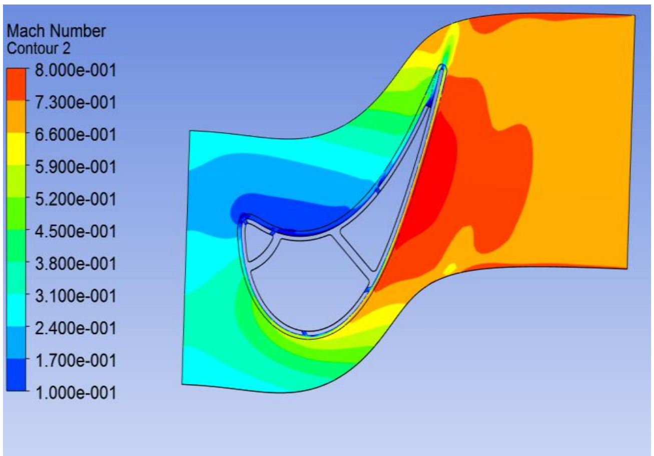


Fig. 9 Temperature and Mach number distributions in the plane of symmetry of the perforation channel.

Conclusions

When designing blades with a developed system of convective-film cooling, in the first stage only the calculation of the thermal state of the blade can be used. In this case, the boundary conditions for heat exchange should be corrected for the presence of film cooling. These corrections can be obtained from the solution of model conjugate problems. When using channels with fan-shaped holes, it should be taken into account that at the same pressure and temperature of the cooling air, the cooling air mass flow is higher than in a similar channel with cylindrical holes.

References

[1] Isaev S. A., Leont'ev A. I., and Baranov P. A., "Problems of Simulating tornado-like heat transfer in turbulent flow past a dimpled relief on a narrow channel wall", Journal of Engineering Physics and Thermophysics, Vol. 83, No. 4, 2010

[2] G.N. Xie, B.Sunden, L.Wang, E. Utriainen. ASME Heat Transfer Conferenct, HT2009, Paper no. HT2009-88116.

[3] J. Park, P.R. Desam, P.M.Ligrani / Numerical prediction of the flow structure above dimpled surface in f channel. Numerical Heat Transfer A 45 (2004) 1 - 20.

[4] Zhestkov G.B., Kharkovsky S.V., Shcherbakova E.V., Belov S.V., Muravsky D.V. Modeling and investigation of external heat transfer during convective-film cooling of TVD blade elements. - Report at the All-Russian scientific and technical conference "Aircraft engines of the XXI century". Moscow, FSUE "CIAM them. P.I. Baranova », November 24-27, 2015

[5] Belov S.V., Zhestkov G.B., Muravsky D.V., Kharkovsky S.V., Shcherbakova E.V. Calculation and experimental study of external heat transfer during convective-film cooling of TVD blade elements. - Abstracts for the report at the XV Minsk International Forum on Heat and Mass Transfer. Minsk, 2016, T.1, pp.32-35.

[6] Menter, FR, Langtry, RB, Likki, SR, Suzen, YB, Huang, PG, and Völker, S., "A Correlation based Transition Model using Local Variables Part 1- Model Formulation", ASME-GT2004-53452, ASME TURBO EXPO 2004, Vienna, Austria.

[7] Langtry, RB, Menter, FR, Likki, SR, Suzen, YB, Huang, PG, and Völker, S., "A Correlation based Transition Model using Local Variables Part 2 - Test Cases and Industrial Applications", ASME-GT2004 - 53454, ASME TURBO EXPO 2004, Vienna, Austria.

[8] Langtry, R.B., Menter, F.R., "Transition Modeling for General CFD Applications in Aeronautics" AIAA paper 2005-522, 2005.

[9] Schlichting G. The theory of the boundary layer. Moscow: Nauka, 1969, 744 p..

Copyright Statement

Contact Author Email Address

Mail to: kharkovski@ciam.ru

zhestkov_g@ciam.ru

Zhestkov_gb@mail.ru