ISSN 0040-6015, Thermal Engineering, 2013, Vol. 60, No. 2, pp. 92–97. © Pleiades Publishing, Inc., 2013.
Original Russian Text © A.A. Inozemtsev, A.S. Tikhonov, C.I. Sendyurev, N.Yu. Samokhvalov, 2013, published in Teploenergetika.

Achieving Better Cooling of Turbine Blades Using Numerical Simulation Methods

A. A. Inozemtsev, A. S. Tikhonov, C. I. Sendyurev, and N. Yu. Samokhvalov

Aviadvigatel', Komsomol'skii pr. 93, Perm, Perm krai, 614990 Russia

Abstract—A new design of the first-stage nozzle vane for the turbine of a prospective gas-turbine engine is considered. The blade's thermal state is numerically simulated in conjugate statement using the ANSYS CFX 13.0 software package. Critical locations in the blade design are determined from the distribution of heat fluxes, and measures aimed at achieving more efficient cooling are analyzed. Essentially lower (by 50–100°C) maximal temperature of metal has been achieved owing to the results of the performed work.

Keywords: numerical simulation, conjugate statement, film cooling, nozzle vane, turbine

DOI: 10.1134/S0040601513020031

Increased life, enhanced reliability, and better fuel efficiency are the most important conditions for achieving competitiveness of gas turbine engines. Increase of gas temperature upstream of the turbine T_g^*, which is required for making the gas-turbine engine more efficient, entails more severe conditions under which the components of its hot part operate and generates the need of carrying out comprehensive substantiation in designing.

The local temperature of gas at the turbine inlet T_g^* may reach—with due regard to circumferential nonuniformity—2500 K. Under such conditions, the greatest difficulties arise with organizing reliable cooling of the first-stage nozzle vanes in the high-pressure turbine (HPT). High levels of temperatures and gas velocities, and the curvature of the vane's cooled surfaces are factors that negatively affect the reliability of its cooling and in the final analysis may lead to the occurrence of cracks and burnouts.

In designing turbine blades, numerical analysis of gas dynamics and the thermal state of blades [especially perforated first-stage nozzle vanes (NV1)] is usually carried separately because solution of a conjugate problem (gas dynamics + heat transfer) involves a very high computation effort.

The objectives of this work are to carry out numerical simulation of the thermal state of NV1 used in the turbine of an aircraft gas-turbine engine (GTE) in conjugate statement, to determine the critical locations in the design of this vane, to work out design measures for achieving better cooling of the vane, and to compare the efficiency of these measures.

STATEMENT OF THE PROBLEM

A new three-cavity NV1 of an HPT (Fig. 1) fitted with a convective-film cooling system is the subject of the study. Such a system is composed of three deflectors organized by placing inserts in the first, second, and third cavities of the vane and by making perforation over the vane profile. Air from the deflectors enters into the vane wall cavities through the rows of holes. Air to the first cavity is supplied from above (through the outer ring), and air to the second and third cavities is supplied from the bottom (through the inner blade). The second and third cavities have longitudinal fins inside them for enhancing convective heat transfer and for increasing the stiffness of the vane suction side. In addition, the second and third cavities are subdivided into three parts along the height. The partitions make the vane stiffer and hold the deflector inside the vane's bucket.

The vane's first cavity is subdivided by vertical partitions into two more cavities a high-pressure cavity (on the pressure side) and a low-pressure cavity (on the suction side). With such subdivision, it becomes possible to obtain the optimal parameters of blowout and, accordingly, to set up film on both the vane's pressure and suction sides and also to obtain guaranteed pressure difference at the entrance edge.

The numerical analysis was carried out using a 3D calculation procedure central to which was solution of the unsteady system of Navier–Stokes' Reynolds-averaged equations using the finite volume method with an implicit integration algorithm (ANSYS CFX 13.0). To this end, geometrical models of the NV1 and of the calculated flow region (which includes also the cavities for supplying cooling air to the NV1 from the combustion chamber's (CC) external loop) were constructed in the 3D simulation software package (NX6).