Verification of a Numerical Model of a Two-Stage HPT of a Modern GTE for Civil Aviation

Grigorii Popov*, Vasilii Zubanov, Evgenii Goriachkin, Anastasia Scherban, and A. A. Shvyrev

Samara National Research University, Samara, Russia
UEC-AVIADVIGATEL, Perm, Russia
*Correspondence: grishatty@gmail.com

***Abstract*—The paper describes the results of the first step of the research team of the Department of Theory of Aircraft Engines to modernize the working process of a cooled axial two-stage high pressure turbine. The paper describes 2 numerical models of the turbine. The first one is with a small number of finite volumes. It is relatively accurate, but requires moderate computer resources to obtain results. The application of this model is planned for the optimisation process. The second one is with a large number of finite volumes. It is expected to have lower error rate and high computational cost. It will be used by the authors for verification calculations to check the found optimal variants and to choose the final variant. The created numerical models are verified by the available experimental data. The paper substantiates the choice of the number of finite volumes in the annular section of the blade passage, as well as their distribution along the blade height. It is shown that the integral parameters of the turbine obtained in the calculation lie in the scatter field of experimental data.**

***Keywords*—axial turbine, numerical simulation, verification, optimization, finite volumes, accuracy**

I. INTRODUCTION

Modern turbines in gas turbine aircraft engines operate at high inlet gas temperatures. This can reach up to 2000K. This temperature exceeds the melting point of blade materials and the blades are cooled intensively to ensure their performance [1, 2]. Nowadays open cooling systems are applied in turbines where the turbine is cooled by "cold air". It flows through the internal channels of the blade and comes out to the flow path through the holes on the surface of the blade or near the trailing edge.

Blowing coolant into the flow path significantly changes the flow pattern in the channels. For the qualitative design of new and upgraded cooled turbines, the interaction process between the main flow and the coolant jets must be modelled. However, with CFD modelling this would lead to a significant increase in the size of the model and would be resource-intensive and time-consuming to compute. As a result, the time for finding a design solution will increase significantly up to unacceptable values. The use of mathematical optimisation methods in this case will also not be effective, as their application requires thousands of runs to the calculation model, which will also require a huge amount of time.

Successful design of cooled turbines requires the development of existing CFD modelling methods. It is necessary to find model settings that not only require fewer computational resources, but also provide sufficient accuracy of the results. Finding such CFD settings is the goal of this paper. It should be noted, however, that the problem is solved from the perspective of a practice engineer who can change the available settings (turbulence models and volume grid settings), but cannot modify the model algorithms.

Cooling can be modelled both directly, by reproducing the geometry of the blade internal channels and directly modelling their processes [3-5, 11, 12]. Obviously, the approach is resource intensive, and it is unlikely that the model dimensionality can be significantly simplified with such an approach. However, the authors do not dismiss this model, as it has greater validity due to fewer simplifications. The authors plan to use it for reference calculations of the most promising turbine variants found with simpler models.

For quick search optimisation calculations, the authors use a model in which the internal blade channels are not modelled and the coolant discharge is simulated as point sources of mass flow rate (Fig. 1). In this case the mesh size is practically not increased in comparison with the model without cooling and the time of preparation of numerical model is considerably reduced. Such models are useful in the design and computational development phases when multiple turbine variants need to be calculated and compared.

Selection of the most rational settings of a numerical model of a cooled turbine was performed using an example of a 2-stage cooled axial turbine developed at UEC Aviadvigatel [6]. The turbine has gas expansion ratio π_t^*=4.76. Fig. 2 shows relative flow rates of cooling air at the turbine inlet relative to the flow rate at the inlet to the first nozzle guide vane of the turbine.

Manuscript received August 23, 2022 ; revised September 16, 2022; accepted November 25, 2022.

doi: 10.18178/ijmerr.12.2.78-83

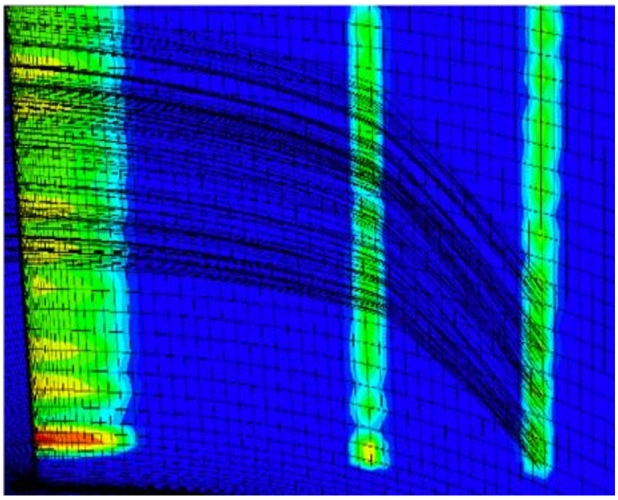

Figure 1. Example of a numerical model of a cooled turbine in which the coolant is blown out of the grid cells.

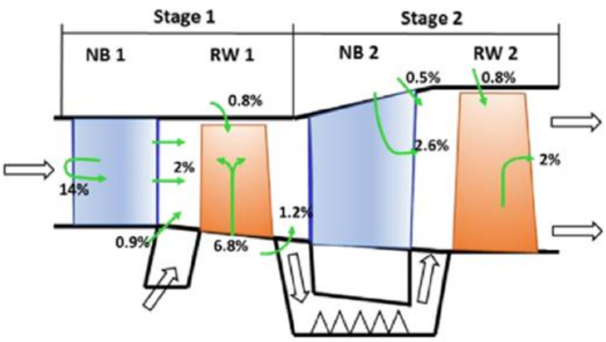

Figure 2. Flow path diagram of the turbine in question.

II. NUMERICAL TURBINE MODEL

A numerical model of the turbine working process was created with Numeca Fine/Turbo (Fig. 3). This program is based on the solution of the averaged Navier-Stokes equations (RANS). The choice of such a tool is due to the fact that today it is the most accurate calculation method for describing hydrogeodynamic processes (because it describes the flow with minimal assumptions).

The created mathematical model takes into account only the gas-dynamic processes in the blade channels of the turbine (including the mixing of the main flow and the coolant), but does not take into account heat transfer and strength issues (these issues are beyond the scope of the study).

Modelling was performed in the stationary formulation with the condition of cyclic symmetry. The model takes into account the presence of an inter-disk cavity between the nozzle guided vanes and the rotor wheel of the 1st stage, as well as the cavity of the labyrinth seal under the NGV of the 2nd stage. The model takes into account the radial clearance on the turbine blades.

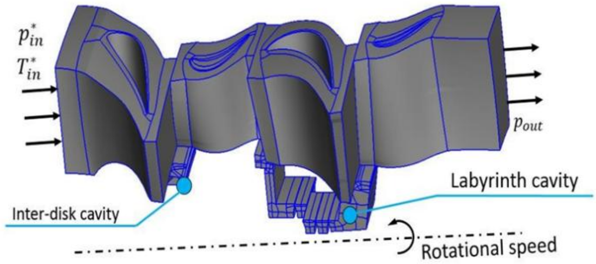

Figure 3. Numerical model of the turbine working process.

An ideal gas with combustion product properties of kerosene was used as the working fluid. Turbulence model is Spalart-Allmaras.

At the turbine inlet, the average values of total pressure and temperature have been set. The direction of the flow angle at the turbine inlet is axial. At the turbine outlet the static pressure at the hub radius was set. Pressures at other radiuses were calculated by radial equilibrium equation. Static pressure value at the turbine outlet was set according to required total pressure ratio. For data transfer between the NGV and RW areas, a Full Non Matching Mixing Plane type interface was used in the Numeca Fine/Turbo software package. The numerical values of the boundary conditions were taken from the data of thermodynamic calculation and experimental data. The authors cannot provide more detailed initial data on the boundary conditions used at the request of the customer of the work.

The parameters of the calculated finite volume grid are shown in Table I.

TABLE I. CALCULATION GRID PARAMETERS ADOPTED FOR MODELLING THE TURBINE IN QUESTION.

Domain	Total number of elements, mln.	Layers by height	Cells per layer, ths.	ER	MR
NGV 1	2.8	105	27	1.2	910
RW1	3.3	121	27.3	1.2	376
NGV 2	2.8	109	25.6	1.2	287
RW2	3.5	117	29.9	1.2	636
Inter-disk cavity	0.5	-	-	1.8	62
Labyrinth	2.7	-	-	1.8	107

The number and height distribution of grid layers were controlled using the following parameters (Table I).

Expansion ratio (ER) – shows how many times the cell height of the next grid layer is smaller than the previous one; Maximum aspect ratio (MR) is the relative height of cells in the flow core. Represents the maximum height of cells in the channel (on the average radius) relative to the size of the first element (Fig. 4).

Modelling of cooling air discharge was performed by blowing additional working fluid out of the calculation grid cells (Fig. 5, a). The grid in the area of the holes was adopted based on the experience of similar calculations by the authors. Unfortunately, the influence of cell size on the calculation results cannot be studied due to the lack of a detailed distribution of parameters by profile. Data were available only in the control sections at the inlet and outlet of the stage.

Positioning of cooling air exhaust rows was performed in relative terms by variables Bi, Rhi and Rsi (Fig. 5, b), where Bi is the relative coordinate of the i-th cooling air exhaust row along the blade surface; Rhi and Rsi are the relative height at which the i-th cooling air exhaust row starts and ends respectively.

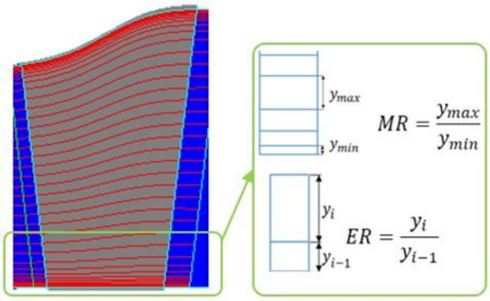

Figure 4. Schematic diagram for determining ER and MR parameters.

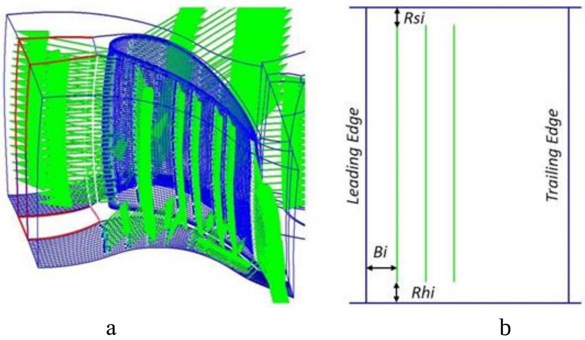

Figure 5. Description of cooling air exhaust from the computational grid cells.

Angles of direction of cooling air blows were set relative to the surface of the design grid at the hole locations.

III. Verification of the Numerical Turbine Model

Aviadvigatel provided the authors' team with the results of experimental studies of the initial version of the HPT. In particular, the distribution of the total pressure over the altitude at the inlet and outlet of the experimental set-up; the altitude distribution of the total temperature at the inlet and outlet of the experimental unit; the altitude distribution of total pressure at the inlet to NGV1 and NGV2 were available (Fig. 6). The pressure was measured by special radial pressure receivers placed in the flow part.

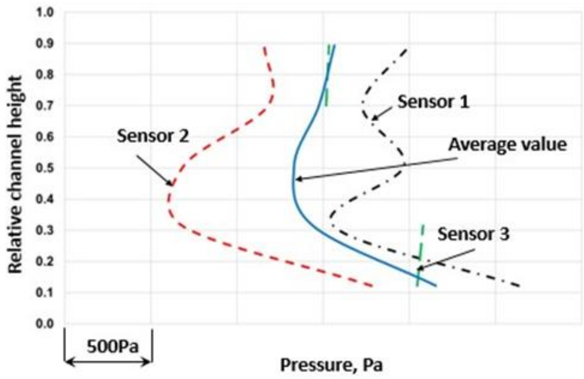

Figure 6. Example of available initial data for turbine numerical model verification.

The values of the specified parameters were known at 5 points evenly spaced along the height. For each height, the parameter values were known at three positions around the circumference of the canal.

The minimum, maximum and nominal values were determined for each parameter. On the basis of these values, the scatter of the measured parameters during the experiment was determined.

Using the scatter of measured pressures and temperatures, the possible range of turbine efficiency and power values from the experimental results was determined. The power value varied between −2.88% and 3.17% compared to the average value, and the efficiency between −2.25% and +2.53%.

From the results of the numerical simulation, as well as the processing of the experimental results, the minimum, average and maximum values of the turbine parameters were calculated at five radial positions, equally distributed over the blade height. An example of a comparison between the calculated and experimental distributions is shown in Fig. 7 and 8.

Comparison of the calculated and experimental distributions of the total temperature over height showed that qualitatively the character of their distributions is slightly different, nevertheless the quantitative difference between the calculated and experimental values does not exceed 3%. The difference in the characters of the curves is explained by the fact that the numerical calculation allows us to consider many points along the height of the channel. Experimental data-only 5 and the distance between them is high. And therefore such density of devices is not enough for receiving a detailed radial diagram. It can be seen that although the characters of the curves are different, the values at the control points are close.

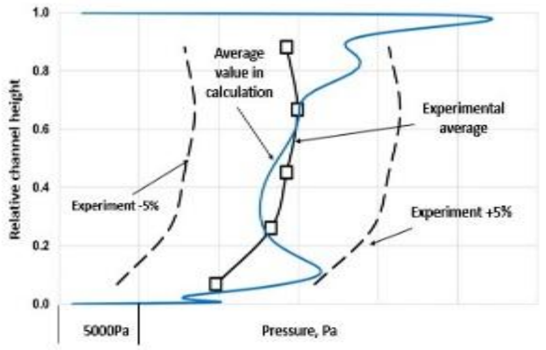

Figure 7. Comparisons of Calculated and Experimental Distributions of Total Pressure Along the Relative Blade Height.

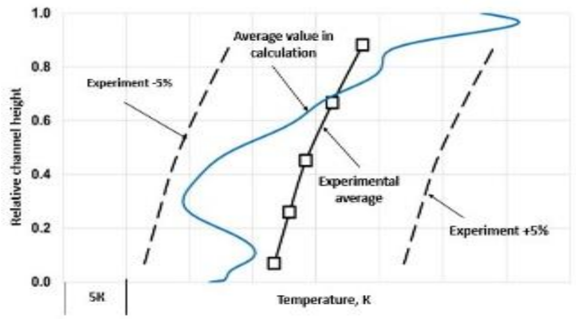

Figure 8. Comparisons of Calculated and Experimental Distributions of Total Temperature Along the Relative Blade Height.

Comparison of the calculated and experimental height distributions of total pressure showed that they are in a qualitative good agreement with each other. A summary of obtained deviations of averaged calculated and experimental values is given in Table II.

TABLE II. DIFFERENCES BETWEEN CALCULATED AND EXPERIMENTAL VALUES.

Comparable parameter	Deviation experimental data
Averaged power value	0.5 %
Averaged efficiency value	1.5 %
Flow-rate - averaged value of total pressures across sections	< 2.2 %
Flow-rate - averaged value of total temperatures across sections	< 1 %
Average pressure at the i-th radius	< 1 %
Average temperature at the i-th radius	< 3 %

IV. SEARCHING FOR GRID SETTINGS FOR OPTIMISATION CALCULATIONS

The numerical model described in the previous section had 11 million finite volumes. Calculation of one point on a modern personal computer with 6 processors took 3 hours. According to the authors' experience, to find an optimum with the help of Indirect Optimisation on the basis of Self-Organization (IOSO) algorithm [7-9] with the planned number of variables to be varied, 4000 calls to the computational model will be required, which corresponds to the calculation time of 12000 hours (500 days). This time is excessive even taking into account the involvement of additional computers in the calculation.

For this reason, the computational grid of the numerical model of the turbine was reduced (it allowed to reduce the time of calculation of one point by 40 minutes (by 4.5 times)). This grid will be further used in the optimisation cycle. And the initial numerical model described in the previous section will be used to carry out control calculations of the best options in order to refine their performance with a better and more detailed calculation model.

The following steps were taken to simplify the computational grid (and reduce the computation time): the labyrinth seal cavity domain was excluded from the computational model, and instead the air intakes and outflows from the computational grid cells were specified (Fig. 9); the density of elements in the computational grid was reduced.

In order to validate the choice of design grid settings for the optimisation calculation, mesh studies were carried out. During the mesh investigations, the mesh density settings were changed in circumferential direction as well as along the height.

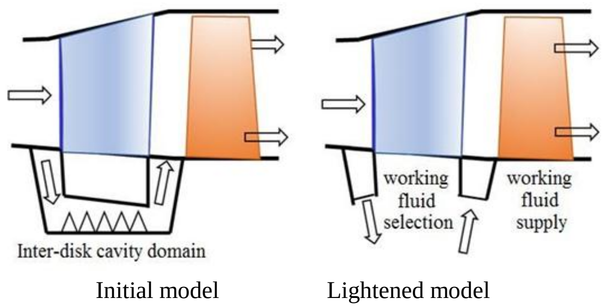

Figure 9. Labyrinth seal cavity replacement for air selection and supply.

In the experience of the authors [10] the ER parameter affects the total number of elements in the computational mesh significantly less than the MR parameter. Therefore, the ER parameter was not changed during the mesh investigations and ER=1.2 was assumed in all computational meshes. And the change in the number of elements in the computational meshes by height was performed only by the MR parameter.

The number of elements in the circumferential direction varied from 10 to 43 thousand per mesh layer (in a conical section perpendicular to the radius). The total number of elements in the numerical model of the two-stage turbine varied from 8.3 to 28.7 million.

The MR parameter varied from 250 to 2000. The number of elements in the numerical model of the two-stage turbine varied from 15.5 to 25.1 million.

The grid convergence analysis considered the effects of the number of grid elements on the integral parameters of the turbine in the Fig. 10 and the distribution of energy loss coefficients in the RW and NGV over the height of the flow path in the Fig. 11. As an example, Fig. 11 shows the effect of the height distribution of the loss factor for the RW 1st stage and the NGV 2nd stage at different values of the MR parameter.

The integral parameters of the turbine were determined using the same methodology as that used for the experimental results. The calculations show that the number of elements in one grid layer affects the integral parameters of the turbine, but has almost no effect on the distribution of the parameters over the height of the flow path.

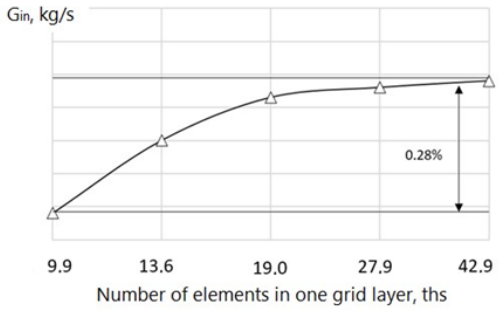

Figure 10. Influence of the number of elements in one grid layer on the value of working fluid flow rate through the turbine.

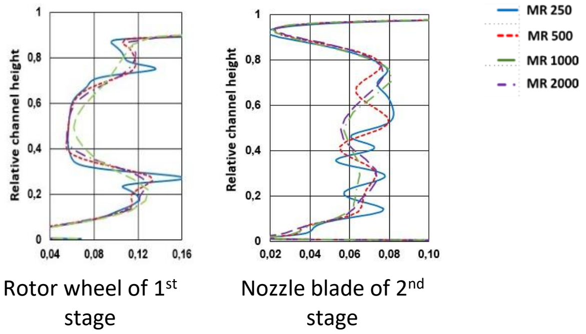

Figure 11. Effect of the MR parameter on the height distribution of the loss coefficient.

In order to quantitatively assess the influence of the number of elements in one mesh layer, the dependencies of the integral parameters of the turbine on the number of elements were plotted. For example, Fig. 10 shows the effect of the number of elements on the calculated value of flow rate through the turbine.

Fig. 11 shows that by increasing the number of elements from 10 to 19 thousand elements in the grid layer, the working fluid flow rate changes by 0.25% (which is comparable to the error in the experimental determination of the flow rate). Further refinement of the mesh leads to only a small change in the mass flow rate. Thus, for verification calculations it is reasonable to use mesh model with the number of elements in one layer not less than 19 thousand, as just in this case mesh convergence is observed (further mesh refinement has little effect on quantitative results).

To perform optimisation calculations, it is suggested to use a mesh with 9.9 thousand elements in one layer as quantitative estimation of integral parameters of such model does not significantly differ from estimation by exact model with dense mesh.

Fig. 11 shows that when the MR parameter is changed, the turbine parameter height distribution changes qualitatively. At values of parameter MR=1000 and higher there is smoothing of peaks of loss coefficient (related to secondary losses) along the height. This suggests that meshes with MR values greater than 1000 may not have sufficient density to correctly find the optimum configuration. Thus, for both verification and optimisation calculations it is reasonable to use meshes with MR values equal 1000.

V. Conclusion

This paper presents the results of work on creating a CFD model of an Axial Cooled Two Stage Turbine and its verification with the available experimental data. The model is planned to be used in future optimisation calculations to find the best turbine configuration. Therefore, a great deal of attention was paid to finding configurations that would provide a low computer resource requirement but would also give a good match to the experimental data.

In the course of research, it was found that the created numerical model allows to find turbine capacity with an accuracy of 0.5%, efficiency with an accuracy of 1.5%, values of total pressure and temperature in the flow path with an accuracy of not less than 2%. It is established that in order to achieve such parameters the number of elements in one layer of finite volume mesh (B2B mesh) must be 10 thousand, MR criterion value for the mesh must be at least 1000, ER criterion equal to 1.2.

Conflict of Interest

The authors declare no conflict of interest.

Author's Contribution

All authors made an equal contribution to the preparation of the publication.

Nomenclature

π_t^*	gas expansion ratio in the turbine;
α_{exit}	turbine flow exit angle.
GTE	gas turbine engine;
HPT	high pressure turbine;
NGV	nozzle guide vane;
RW	rotor wheel;
ER	Mesh cells Expansion ratio;
MR	Mesh cell maximum aspect ratio.

References

[1] M. P. Boyce, Gas Turbine Engineering Handbook, 2006. 10.1016/B978-0-7506-7846-9.X5000-7.

[2] A. Aminaee, *Gas Turbine Design, Components and System Design Integration*, Springer, 2018. https://doi.org/10.1007/978-3-319-58378-5

[3] K. Vinogradov, G. Kretinin, K. Otryahina, R. Didenko, D. Karelin, et al., "Robust optimisation of the HPT blade cooling and aerodynamic efficiency," in *Proc. the ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition*, vol. 2C, Turbomachinery, 2016.

[4] K. Ho, J. S. Liu, T. Elliott, and B. Aguilar, "Conjugate heat transfer analysis for Gas turbine film-cooled blade," in *Proc. the ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition*, vol. 5A, Heat Transfer, 2016.

[5] J. Liu, M. C. Morris, M. F. Malak, R. M. Mathison, M. G. Dunn, "Comparison of 3D unsteady transient conjugate heat transfer analysis on a high pressure cooled turbine stage with experimental data," in *Proc. the ASME Turbo Expo 2017: Turbomachinery Technical Conference and Exposition*, vol. 5A, Heat Transfer, 2017.

[6] JSC UEC-Aviadvigatel, [Online]. Available: https://avid.ru/en/, last accessed 2022/07/28.

[7] I. N. Egorov, G. V. Kretinin, I. A. Leshchenko, and S. V. Kuptzov, "The main features of IOSO technology usage for multi-objective design optimization," in *Proc. 10th AIAA/ISSMO Multidisciplinary Analysis and Optimisation Conference*, 2004.

[8] I. N. Egorov, G. V. Kretinin, I. A. Leshchenko, and S. V. Kuptzov, "Multi-objective robust optimisation of air engine using IOSO technology," in *Proc. the ASME Turbo Expo 2004: Power for Land, Sea, and Air*, vol. 2, Turbo Expo 2004, GT2004-53504, 2004, pp. 157-163.

[9] Sigma Technology. [Online]. Available: https://www.iosotech.com, last accessed 2022/07/28.

[10] G. Popov, V. Matveev, O. Baturin, Y. Novikova, and A. Volkov, "Selection of parameters for blade-to-blade finite-volume mesh for CFD simulation of axial turbines," in *Proc. MATEC Web of Conferences*, vol. 220, 2018.

[11] C. C. Easterby, J. D. Moore, and D. G. Bogard, "CFD evaluation of internal flow effects on turbine blade leading-edge film cooling with shaped hole geometries," in *Proc. the ASME Turbo Expo 2021: Turbomachinery Technical Conference and Exposition*, vol. 5A: Heat Transfer—Combustors*; Film Cooling*. Virtual, Online. June 7–11, 2021. V05AT12A023. ASME

[12] H. Du, D. Xie, W. Chen, Z. Mei, and J. Zhang, "Overall cooling effectiveness on a gas turbine vane with swirl-film cooling," in *Proc. of the ASME Turbo Expo 2019: Turbomachinery Technical Conference and Exposition*, vol. 5A: Heat Transfer. Phoenix, Arizona, USA. June 17–21, 2019.

Popov Grigory, Associate Professor, Department of the Theory of Aircraft Engines of the Samara National Research University named after S.P. Koroleva (34, Moskovskoye shosse, Samara, 443086, Russia). Dipl. eng.-construct. (Samara University, 2010). P.H.d. (Samara University, 2018). Specialist in the field of working processes of gas turbine engines, turbomachines, their multidisciplinary numerical simulation and optimization.

Goriachkin Evgeny is Associate Professor, Department of the Theory of Aircraft Engines of the Samara National Research University named after S.P. Koroleva (34, Moskovskoye shosse, Samara, 443086, Russia). Dipl. master (Samara University, 2014). P.H.d. (Samara University, 2021). Specialist in the field of working processes of gas turbine engines, turbomachines, their multidisciplinary numerical simulation and optimization.

Zubanov Vasily is Assoc. cafe TDLA (Samara University). Dipl. master (Samara University, 2016). P.H.d. (Samara University, 2022). Specialist in the field of working processes of gas turbine engines, turbomachines, their multidisciplinary numerical simulation and optimization.

Scherban Anastasia is Junior Researcher, IC 206 (Engineering Center of Samara University). Dipl. eng. (Samara University, 2018). Specialist in the field of working processes of turbomachines, their numerical simulation.

Shvyrev Andrey is deputy head of the department for aerodynamic design of the department for computational and experimental work on turbines UEC-AVIADVIGATEL. Specialist in the field of gas-dynamic design and heat transfer in GTE turbines.