DOI: https://doi.org/10.15407/pmach2019.03.014

UDC 533.2

NUMERICAL ANALYSIS OF WORKING PROCESSES IN THE BLADE CHANNELS OF THE HIGHLY-LOADED TURBINE OF A MARINE GAS TURBINE ENGINE, USING A REFINED FINITE ELEMENT MODEL

Serhii O. Morhun
serhii.morhun@nuos.edu.ua
ORCID: 0000-0003-2881-7541

Admiral Makarov National University of Shipbuilding,
9, Heroyiv Ukrayiny Ave.,
Mykolaiv, 54025, Ukraine

Issues of designing a single-stage high-loaded turbine of a marine gas turbine engine are considered. The object of our research is the aerodynamic characteristics of a viscous three-dimensional turbulent gas stream flow in the flow path of the turbine under consideration. At this stage, a numerical analysis of working processes in the blade passages of the turbine stage has been carried out. When designing the turbine, it is necessary to take into account the fact that the possibilities of improving the flow path shape by optimizing the shapes of blade passages in plane sections do not meet the requirements for high-loaded turbines. An alternative to this approach is the use of computational gas-dynamic methods in a three-dimensional formulation. Therefore, this paper outlines a method for constructing a refined finite element model of the working fluid flow in the flow path of the single-stage high-loaded high-pressure turbine of a marine gas turbine engine. To solve this problem, a finite-element hexagonal-type mesh was constructed using the three-dimensional Navier-Stokes equations for the case of viscous working fluid flow. The three-dimensional model of the turbine flow path presented in this paper consists of two stator sections and four rotor sections. Each section includes a blade airfoil with upper and lower contours, which simply model the root and shroud shelves. In the course of calculations, such types of boundary conditions as "entrance", "exit" and "wall" were used. At the entrance, the total flow pressure and flow temperature were given. Since the turbine is single-stage, then at the entrance to the computational domain, the flow is directed axially. At the exit from the computational domain, the static pressure was given. On the wall, both the non-slip and slip boundary conditions were also used. Using the developed mathematical model, the fields of Mach numbers, flow velocities, and static pressure in the root and peripheral sections of the turbine flow path are determined. The calculation was carried out in a non-stationary setting with a time step of $1.5974 \cdot 10^{-6}$ s, which corresponds to the angle of rotor rotation, relative to the stator, of 0.09 degrees. The total number of time iterations was 200. The results obtained can be applied to further study the strength of the blading of highly-loaded marine gas turbine engines.

Keywords: *marine gas turbine engine, three-dimensional finite elements, turbine flow path, root and peripheral sections, fields of Mach numbers, flow velocities and pressure.*

Relevance of the topic

In the context of introducing energy-saving technologies in the maritime transport of this country, the problems of reducing the time needed to design, manufacture, refine and modernize marine gas turbine engines are relevant. The possibilities of improving their flow paths by using empirical approaches, optimizing the shapes of blade passages in plane sections are almost exhausted. Therefore, at present, the role of computational gas-dynamic methods has significantly increased in engineering practice [1–6]. Together with field experiments, they provide an opportunity to investigate the complex influence of various factors on the characteristics of the blading in three-dimensional stationary and non-stationary settings, reduce the cost of manufacturing gas turbine engines (GTE), as well as increase the competitiveness of maritime GTEs being designed and the profitability of their manufacture.

For a long time, in the design of the flow paths of turbomachines at different design stages, fairly simple gas flow models have been used: one-dimensional, quasi-two-dimensional, two-dimensional (plane and axisymmetric) [7–9]. The emergence of a three-dimensional viscous model of the working fluid flow through turbomachine blading and the numerical solution to this problem, together with a qualitative increase in the capabilities of computing technology, greatly expanded the capabilities of designers, and made it possible to proceed to solving the optimal spatial design of turbomachine blading [9]. The accuracy and versatility of this model, the possibility of its use, both for creating individual blade rows and for the whole blading, made it possible to abandon the use of simpler one-dimensional and two-dimensional models of working fluid flows [7, 8]. When conducting a numerical experiment, an important role is played by the model of turbulence. Thus, in [3], three models of turbulence were tested in calculating turbulent flow and heat transfer on the surfaces of straight nozzle array blades. The models used are Spalart-Allmaras (*S–A*), Wilcox (k–ω) and Menter (*SST*). When the *S–A* and SST models were used for calculations, obtained were the flow patterns that are close to each other and to the experimental ones at the leading edges of blades.

Based on the analysis of the above sources, it can be concluded that at the moment there are still a number of unsolved problems regarding the improvement of maritime GTE turbine bladings through numerical methods. Therefore, the aim of the work is to study the parameters of flow in the blade passages of the highly-loaded single-stage high-pressure turbine of a maritime gas turbine engine, using a three-dimensional mathematical model of viscous working fluid flow.

Formulation of the Problem

The object of our research is the aerodynamic characteristics of a viscous three-dimensional turbulent gas stream flow in the flow path of the turbine under consideration.

As the stationary coordinate system, the Cartesian right-handed *xyz* coordinate system with its center at point *O* located on the GTE axis is taken. The *z* axis is perpendicular to the GTE axis, and the *x* axis coincides with this axis. The rotating coordinate system rotates together with the turbine rotor with a constant angular speed, Ω, equal to the rotor rotational speed.

The state of the gas at any point in the region under consideration is determined by the following parameters: pressure (p); temperature (T); components of the flow velocity vector: u, v, w. In addition, to describe the properties of a viscous gas being compressed, it is necessary to know its density ρ and viscosity μ, which can be calculated using the above parameters. Thus, the state of any point in a viscous gas flow is given by five variables. Accordingly, a system of five equations is needed to determine the unknown variables. This system consists of:

– the Navier – Stokes-like motion equation [8]

$$\begin{aligned}
&\rho\cdot\frac{\partial u}{\partial t}+\rho\cdot u\frac{\partial u}{\partial x}+\rho\cdot v\frac{\partial u}{\partial y}+\rho\cdot w\frac{\partial u}{\partial z}=\rho\cdot X-\frac{\partial p}{\partial x}+2\frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial x}\right)+\\
&+\frac{\partial}{\partial y}\left[\mu\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)\right]+\frac{\partial}{\partial z}\left[\mu\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)\right]-\frac{2}{3}\frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)\right];\\
&\rho\frac{\partial v}{\partial t}+\rho\cdot u\frac{\partial v}{\partial x}+\rho\cdot v\frac{\partial v}{\partial y}+\rho\cdot w\frac{\partial v}{\partial z}=\rho\cdot Y-\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)\right]+\\
&+2\frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left[\mu\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)\right]-\frac{2}{3}\frac{\partial}{\partial y}\left[\mu\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)\right];\\
&\rho\frac{\partial w}{\partial t}+\rho\cdot u\frac{\partial w}{\partial x}+\rho\cdot v\frac{\partial w}{\partial y}+\rho\cdot w\frac{\partial w}{\partial z}=\rho\cdot Z-\frac{\partial p}{\partial z}+\frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)\right]+\\
&+\frac{\partial}{\partial y}\left[\mu\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)\right]+2\frac{\partial}{\partial z}\left[\mu\frac{\partial w}{\partial y}\right]-\frac{2}{3}\frac{\partial}{\partial z}\left[\mu\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)\right]
\end{aligned}, \quad (1)$$

where ρ is the gas density, kg/m3; p is the gas pressure, Pa; μ is the dynamic gas viscosity coefficient;

– energy equation [8]

$$\rho\left(\frac{\partial T}{\partial t}+u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right)=\left[\frac{\partial p}{\partial t}+u\frac{\partial p}{\partial x}+v\frac{\partial p}{\partial y}+w\frac{\partial p}{\partial z}\right]+\left[\frac{\partial}{\partial x}\left(\lambda\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(\lambda\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(\lambda\frac{\partial T}{\partial z}\right)\right]+\mu\cdot\Phi, \quad (2)$$

where λ is the thermal conductivity coefficient, W/m·K; T is the gas temperature; Φ is the dissipative function [8];

$$\Phi=2\left[\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial w}{\partial z}\right)^2\right]=\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)^2+\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right)^2+\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)^2-\frac{2}{3}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right), \quad (3)$$

– continuity equation [8]

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0. \quad (4)$$

To describe the dependence of density on temperature and pressure in most cases, the well-known equation of state of an ideal gas is sufficient [1]

$$\rho = \frac{p}{RT}. \quad (5)$$

The dynamic viscosity coefficient is determined as follows [7, 8]:

$$\mu = C_{\mu}\rho\frac{k}{\varepsilon}, \quad (6)$$

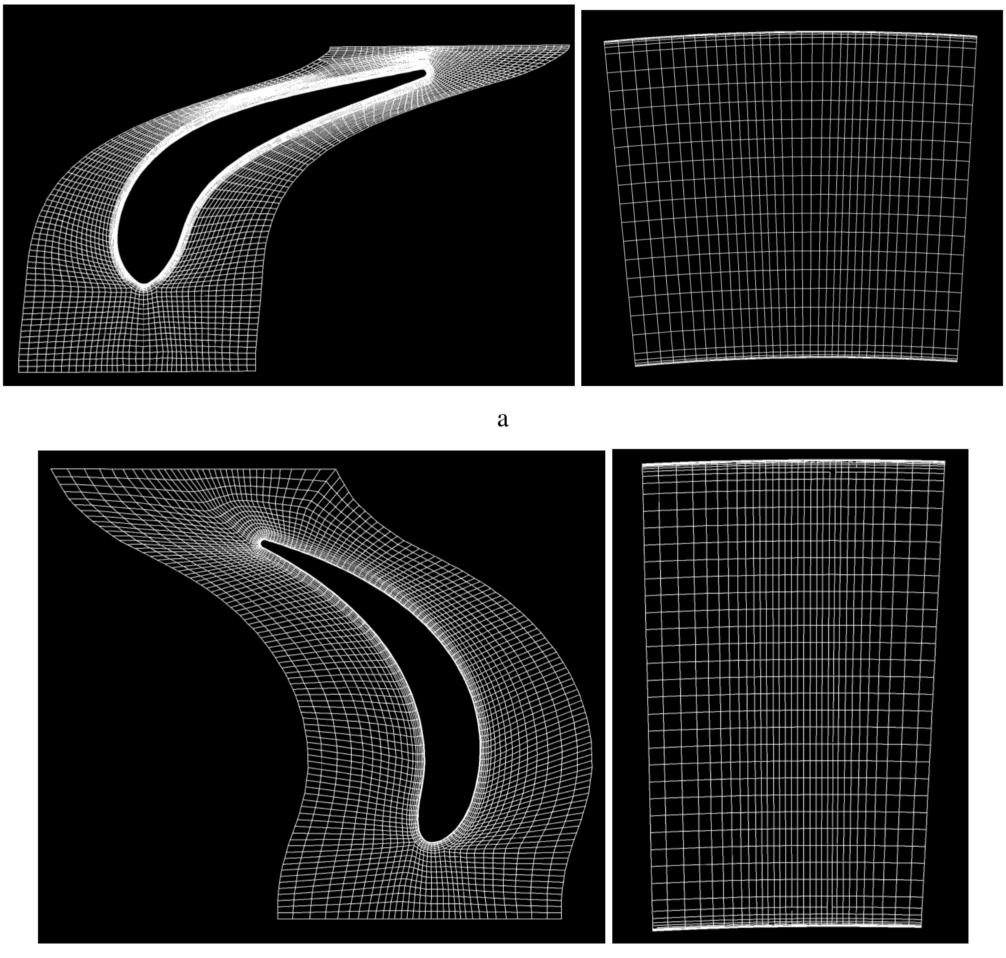
where C_{μ} is a constant; k is the kinetic energy of turbulence; ε is the turbulence dissipation coefficient.

There are no analytical solutions to equations (1–4). To obtain solutions for the case of real gas flow in a turbine, it is necessary to use numerical methods, namely, the finite volume method.

Solution to the Problem

In this work, to solve the problem, a finite-element hexagonal-type mesh was constructed (Fig. 1).

Boundary conditions. To numerically solve the above system of equations (1–5), using a hexagonal finite element mesh, it is necessary to set up boundary conditions. In this case, such types of boundary conditions as "entrance", "exit" and "wall" are considered (Fig. 2).



a

b

Fig. 1. Hexagonal finite element mesh for blade passages:
a – stator; b – rotor

"Entrance". This is the boundary through which the flow "flows" into the domain under consideration. At the entrance, the boundary conditions will be: the total flow pressure and the flow temperature. Since the turbine is single-stage, then at the entrance to the computational domain, the flow is directed axially.

"Exit". At the exit from the computational domain, a static pressure is specified, whose average value over the exit boundary surface is determined as follows:

$$p_s = \frac{1}{A}\int_S p_n dA \ , \qquad (7)$$

where p_s is the static pressure at the boundary; p_n is the pressure value in the computational mesh nodes.

"Wall". When calculating the flows in the blades, of great importance is the modeling of the flow structure near the wall (the surface of the blades), since this calculation influences the correctness of calculating the force acting on blades and losses due to friction. To describe the turbulent boundary layer, wall functions are usually used, which are a set of semi-empirical functions that connect the value of the independent variables in the center of the computational cell with the values of the corresponding variables on the wall, and are based on the assumption of Launder and Spalding [10]. In this case, on the wall, both the non-slip and slip boundary conditions were used.

The three-dimensional turbine flow path model presented in this paper consists of two stator sections and four rotor sections (Fig. 3). Each section includes a blade airfoil with upper and lower contours, which simply model the root and shroud shelves.

The numerical solution to the system of equations (1–5), taking into account boundary conditions (6, 7), allows determining the fields of Mach numbers, static pressure, and flow velocities in the root and peripheral sections of stator and rotor blades. The calculation is carried out in a non-stationary setting with a time step of $1.5974 \cdot 10^{-6}$ s, which corresponds to the angle of rotation of the rotor, relative to the stator, of 0.09 degrees. The total number of time iterations is 200.

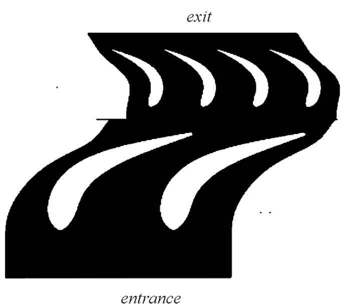


Fig. 2. Conditions at the boundaries of the computational domain

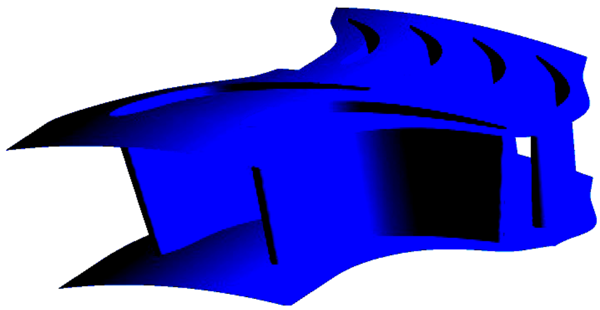


Fig. 3. Three-dimensional model of the flow path of the turbine under consideration

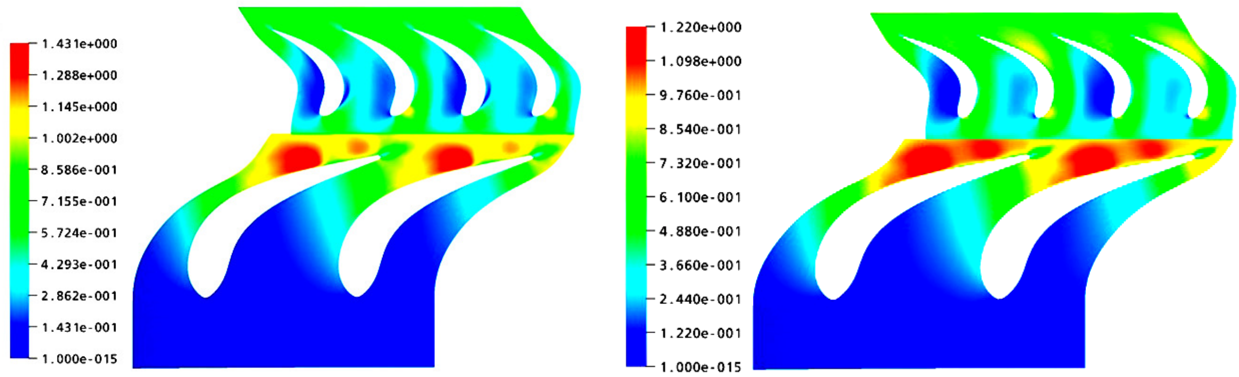
Main Results and their Analysis

The power of the turbine under study is 25 MW. The temperature of the working fluid at the stage entrance is 1583 K; the total pressure at the stage entrance is 0.7578 MPa; the rotor speed is 9 390 rev/min.

The spatial structure of the flow is clearly represented by a number of axisymmetric sections in the nozzle and working rims of the turbine. Fig. 4 shows the field of Mach numbers in the root and peripheral blade sections.

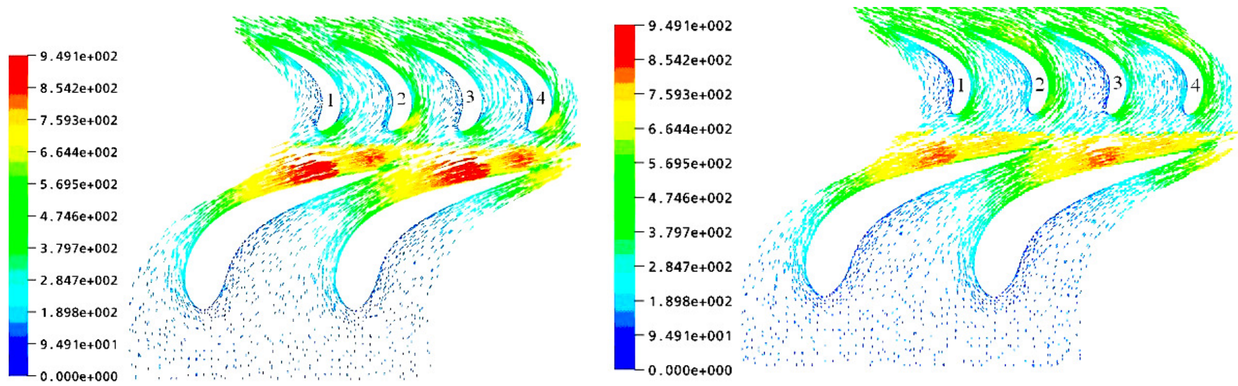
The flow at the nozzle diaphragm inlet has a subsonic velocity and axial direction. In the lattice, the flow accelerates to a supersonic velocity value. The sound line M=1 completely overlaps the blade passage in the root section. At the periphery, the flow velocity at the nozzle lattice exit does not exceed M=0.9. From the side of the back, at the blade passage exit, there is a local jump in flow velocity along the entire height of the nozzle lattice, with the flow velocity jump reaching the value M=1.43 at the root, after which the boundary layer separation occurs. Further, the flow decelerates in the shock wave and enters the working lattice with a transonic velocity value. The velocity gradient in height at the nozzle lattice exit is caused by an increase in the section area of the blade passage exit from the root to the periphery. In the working lattice, the flow accelerates, but does not reach the speed of sound. Due to the twisting of working blades, the cross-sectional area at the blade passage exit decreases from the root of blades to the periphery, which is why, in relative motion, the flow has a radial velocity gradient.

The vector field of velocities (Fig. 5) shows that the flow flows around blades 2 and 4 without impacts and boundary layer separations. On the backs of blades 1 and 3, which fall into the zone of action of the wake behind the edges of nozzle blades, the boundary layer separates and creates a zone of reverse flows, and from the side of the blade pressure surface, there are stagnation regions, most intensely expressed with uneven field parameters at the turbine inlet.



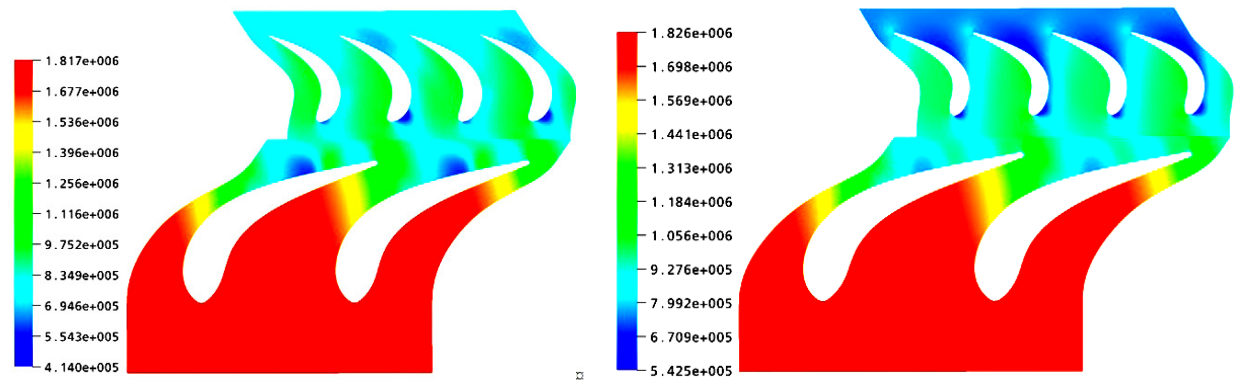
a b

Fig. 4. Distribution of Mach numbers in turbine flow path sections:
a – root section; b – peripheral section



a b

Fig. 5. Vector velocity field (m/s) in the turbine flow path:
a – root section; b – peripheral section



a b

Fig. 6. Distribution of static pressure (Pa) in the turbine flow path:
a – root section; b – peripheral section

The analysis of Fig. 5 indicates the presence of local velocity jumps from the side of the back of nozzle blades, most distinct in the root section (Fig. 5, a). The presence of these velocity jumps leads to the appearance of local pressure jumps, which is confirmed by the information given in Fig. 6.

Conclusion

The analysis performed shows the relevance of the task of improving the blading of axial turbines for marine GTEs through using computational gas dynamics. Using the three-dimensional Navier-Stokes equations for the case of viscous flow of the working fluid, an improved mathematical model based on hexagonal finite elements has been developed. Using this mathematical model, the fields of Mach numbers, velocities and pressures in the root and peripheral sections of the flow path of the single-stage turbine of a maritime gas turbine engine are determined. The results obtained can be applied in the further study of the strength of the blading of the GTE under consideration.

References

1. Kholshchevikov, K. V., Yemin, O. N., & Mitrokhin V. T. (1986). *Teoriya i raschet aviatsionnykh lopatochnykh mashin* [Theory and calculation of aircraft blade machines]. Moscow: Mashinostroyeniye, 432 p. (in Russian).
2. Rizk N. K. & Mongia, H. C. (1990). Three-dimensional combustor performance validation with high-density fuels. *Journal of Propulsion and Power*, vol. 6, no. 5, pp. 660–667. https://doi.org/10.2514/3.23268
3. Menter F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, vol. 32, no. 8, pp. 1598–1605. https://doi.org/10.2514/3.12149
4. Takemilsu, N. (1990). An analytical study of the standard *k-e* model. *Journal of Fluid Engineering*, vol. 112, iss. 2, pp. 192–198. https://doi.org/10.1115/1.2909387
5. Ivanov, M. Ya. & Krupa, V. G. (1993). Computation of 3D viscous cascade flow. *Fluid Dynamics*, vol. 28, iss. 4, pp. 476–484. https://doi.org/10.1007/BF01342682
6. Shang, T. & Epstein, A. H. (1997). Analysis of hot streak effects on turbine rotor heat load. *ASME Journal of Turbomachinery*, vol. 119, iss. 3, pp. 544–553. https://doi.org/10.1115/1.2841156
7. Samarskiy, A. A. & Vabitsiyevich, P. N. (2009). *Vychislitelnaya teploperedacha* [Computational heat transfer]. Moscow: Editorial, 784 p. (in Russian).
8. Sosunov, V. A. & Chepkin, V. M. (2003). *Teoriya, raschet i proyektirovaniye aviatsionnykh dvigateley i energeticheskikh ustanovok* [Theory, calculation and design of aircraft engines and power plants]. Moscow: Moscow Power Engineering University Publ., 677 p. (in Russian).
9. Lecheler, S. (2014). *Numerische stromungsberechnung* [Numerical current calculations]. Munich: Springer Fachmedien Wiesbaden, 199 p. https://doi.org/10.1007/978-3-658-05201-0 (in German).
10. Launder, B .E. & Spalding, D. B. (1994). The numerical computation of turbulent flows. *Computational Methods of Applied Mechanics Engineering*, vol. 3, iss. 2, pp. 269–289. https://doi.org/10.1016/0045-7825(74)90029-2

Received 06 May 2019

Числовий аналіз робочих процесів в міжлопаткових каналах високонавантаженої турбіни суднового газотурбінного двигуна з використанням уточненої скінченноелементної моделі

С. О. Моргун

Національний університет кораблебудування імені адмірала Макарова,
54025, Україна, м. Миколаїв, пр. Героїв України, 9

Розглянуто питання проектування одноступінчатої високонавантаженої турбіни суднового газотурбінного двигуна. Об'єктом дослідження є аеродинамічні характеристики в'язкої тривимірної турбулентної течії газового потоку в проточній частині розглянутої турбіни. Проведено числовий аналіз робочих процесів в міжлопатковому каналі турбінного ступеня. Під час проектування слід враховувати той факт, що можливості удосконалення форми проточної частини за допомогою оптимізації форм лопаткових каналів в плоских перерізах не відповідають вимогам, що ставляться до високонавантажених турбін. Альтернативою такому підходу є застосування методів обчислювальної газової динаміки в тривимірній постановці. Тому в цій роботі викладена методика побудови уточненої скінченноелементної моделі течії робочого тіла в проточній частині одноступінчатої турбіни високого тиску суднового газотурбінного двигуна. Для розв'язання поставленої задачі побудована скінченноелементна сітка гексагонального типу з використанням тривимірних рівнянь Нав'є-Стокса для випадку в'язкої течії робочого тіла. Наведена тривимірна модель проточної частини турбіни складається з двох секцій статора і ро-

тирьох секцій ротора. Секція включає в себе перо лопатки з верхнім і нижнім обводами, що спрощено моделюють кореневу і бандажну полиці. У процесі розрахунків використовувалися такі типи граничних умов, як «вхід», «вихід» і «стінка». На вході було задано повний тиск потоку і температура потоку. Оскільки турбіна є одноступеневою, то на вході в розрахункову область потік спрямований в осьовому напрямку. На виході з розрахункової області задано статичний тиск. Також на стінці використовувалися граничні умови непротікання і прилипання. З використанням розробленої математичної моделі визначено поля чисел Маха, швидкостей потоку і статичного тиску в кореневому і периферійному перерізах проточної частини турбіни. Розрахунок проводився в нестаціонарній постановці з часовим кроком 1,5974·10^{-6} с, що відповідає куту повороту ротора щодо статора на 0,09 градусів. Сумарна кількість тимчасових ітерацій становила 200. Отримані результати можуть бути застосовані під час подальшого дослідження міцності лопаткового апарата високонавантажених суднових газотурбінних двигунів.

***Ключові слова**: судновий газотурбінний двигун, тривимірні скінченні елементи, проточна частина турбіни, кореневий та периферійний перерізи, поля чисел Маха, швидкостей та тиску.*

Література

1. Холщевиков К. В., Емин О. Н., Митрохин В. Т. Теория и расчет авиационных лопаточных машин. М.: Машиностроение, 1986. 432 с.
2. Rizk N. K., Mongia H. C. Three-dimensional combustor performance validation with high-density fuels. *J. Propulsion and Power*. 1990. Vol. 6. No. 5. P. 660–667. https://doi.org/10.2514/3.23268
3. Menter F. R. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA J.* 1994. Vol. 32. No. 8. P. 1598–1605. https://doi.org/10.2514/3.12149
4. Takemilsu N. An analytical study of the standard *k-e* model. *J. Fluid Eng.* 1990. Vol. 112. Iss. 2. P. 192–198. https://doi.org/10.1115/1.2909387
5. Иванов М. Я., Крупа В. Г. Расчет трехмерного течения вязкого газа в прямых решетках профилей. *Изв. РАН. Механика жидкостей и газа*. 1993. № 4. С. 58–68.
6. Shang T., Epstein A. H. Analysis of hot streak effects on turbine rotor heat load. *ASME J. Turbomachinery*. 1997. Vol. 119. Iss. 3. P. 544–553. https://doi.org/10.1115/1.2841156
7. Самарский А. А., Вабициевич П. Н. Вычислительная теплопередача. М.: «Эдиториал», 2009. 784 с.
8. Сосунов В. А., Чепкин В. М. Теория, расчет и проектирование авиационных двигателей и энергетических установок. М.: Моск. энерг. ун-т, 2003. 677 с.
9. Lecheler S. Numerische stromungsberechnung. Munich: Springer Fachmedien Wiesbaden, 2014. 199 p. https://doi.org/10.1007/978-3-658-05201-0
10. Launder B. E., Spalding D. B. The numerical computation of turbulent flows. *Comp. Methods of Appl. Mech. Eng.* 1994. Vol. 3. Iss. 2. P. 269–289. https://doi.org/10.1016/0045-7825(74)90029-2