UDC 621.165
DOI: 10.15587/1729-4061.2026.365474

# DEVELOPMENT A COMPUTATIONALLY EFFICIENT METHOD TO SOLVE DIRECT AND INVERSE AERODYNAMIC PROBLEMS RELATED TO TURBOMACHINES

**Valery Subotovich**
Doctor of Technical Sciences, Senior Researcher
Department of Thermogasdynamics of Power Machines No. 33
Anatolii Pidhornyi Institute of Power Machines and Systems
of the National Academy of Sciences of Ukraine
Komunalnykiv str., 2/10, Kharkiv, Ukraine, 61046
ORCID: https://orcid.org/0000-0002-7051-4758

**Oleksandr Yudin**
*Corresponding author*
Candidate of Technical Sciences*
E-mail: alyudin78@gmail.com
ORCID: https://orcid.org/0000-0001-5098-7796

**Artem Babaiev**
Candidate of Technical Sciences*
ORCID: https://orcid.org/0009-0005-8685-0974

**Valentyn Barannik**
Candidate of Technical Sciences*
ORCID: https://orcid.org/0000-0003-4549-3577

**Olena Avdieieva**
Candidate of Technical Sciences, Associate Professor*
Department of Computer Modeling of Processes and Systems**
ORCID: https://orcid.org/0000-0002-9358-4265
*Department of Turbine Construction**
**National Technical University «Kharkiv Polytechnic Institute»
Kyrpychova str., 2, Kharkiv, Ukraine, 61002

***This study investigates aerodynamic processes in interblade, axial annular, and radial-axial channels in turbomachines. The task addressed relates to the need to improve the computational efficiency in solving direct and inverse aerodynamic problems when analyzing and designing flow parts of turbomachines.***

***A computationally efficient method for solving direct and inverse aerodynamic problems has been proposed, which provides acceptable accuracy for engineering practice at significantly lower computational costs compared to conventional CFD approaches. The results include a devised single mathematical formulation based on Euler equations for a compressible fluid, written in terms of the flow function.***

***The proposed method makes it possible to consider direct and inverse problems within a common mathematical structure. The system of equations is reduced to a single differential equation and a set of algebraic relations while the global flow problem is decomposed into a set of independent problems for individual cross-sections. The inverse problem is stated as a nonlinear optimization problem based on the mass flow rate agreement condition derived from the energy equation. The computational complexity is reduced by using the flow function and the decomposition of the computational domain.***

***The method combines the capabilities of direct analysis and reconstruction of the flow channel geometry based on the predefined aerodynamic characteristics. Verification by experimental data and direct calculation results demonstrated good agreement between the velocity distributions and the reconstructed geometry; the maximum deviations did not exceed 0.3%.***

***The results could be practically applied to preliminary design, parametric optimization, as well as inverse design of the flow parts in turbomachines***

***Keywords: inverse problem; flow function; Euler equation; quasi-three-dimensional flow; flow section; aerodynamic design***

*Received 24.03.2026*
*Received in revised form 04.05.2026*
*Accepted 12.06.2026*
*Published 29.06.2026*

***How to Cite:*** *Subotovich, V., Yudin, O., Babaiev, A., Barannik, V., Avdieieva, O. (2026). Development a computationally efficient method to solve direct and inverse aerodynamic problems related to turbomachines. Eastern-European Journal of Enterprise Technologies, 3 (7 (141)), 71–82. https://doi.org/10.15587/1729-4061.2026.365474*

## 1. Introduction

Turbomachines are the main elements of power plants, aircraft and rocket engines, gas transportation systems, as well as many industrial technological systems. Increasing their efficiency directly affects the reduction of fuel consumption, reduction of emissions of harmful substances, increasing the reliability of equipment, and reducing the cost of energy production. Given the increased requirements for energy efficiency and environmental safety, the task to improve the aerodynamic characteristics of turbomachines remains one of the priority areas of modern energy and transport engineering.

Aerodynamic processes in the flow parts of turbomachines are characterized by a complex spatial structure of the flow. The flow in interblade channels, annular diffusers, transition and exhaust channels is accompanied by significant pressure gradients, secondary flows, separation phenomena, compressibility of the working fluid, and the interaction of individual elements of the flow part. Even minor changes in the geometry of the channels can significantly affect energy losses, efficiency, stability margin, and other operational characteristics of turbomachines. Therefore, reliable prediction of flow parameters and determining the influence of geometric factors on flow characteristics are of significant practical importance.

Current evolution of computer technology has enabled the widespread introduction of numerical methods for studying flows into the processes of analysis and design of turbomachines. The use of computational hydrodynamics makes it possible to obtain detailed information about the flow structure and optimize geometric parameters at the design stage. At the same time, the constant increase in the requirements for the accuracy of calculations leads to an increase in the complexity of mathematical models and an increase in computational costs. This issue is especially acute when conducting parametric studies, during multivariate design, and while solving inverse problems where it is necessary to repeatedly perform flow calculations for different configurations of the flow part.

An important direction in the development of modern design methods is the use of inverse problems, which make it possible to determine the geometric parameters of the flow part from the predefined aerodynamic characteristics. Such approaches open up the possibility of purposeful formation of the flow structure and increasing the efficiency of turbomachines. At the same time, their practical application requires the construction of mathematical models that combine reasonable physical validity with acceptable computational efficiency.

Thus, it is a relevant task to devise methods for aerodynamic analysis and design of turbomachines, capable of providing a combination of physical reliability and high computational efficiency when solving direct and inverse problems.

## 2. Literature review and problem statement

Current methods of aerodynamic analysis and design of flow parts in turbomachines are largely based on the use of mathematical modeling of the working fluid flow. In [1], an algorithm for calculating compressible flows in interblade channels of turbomachines is proposed, which provides the determination of the main flow parameters and velocity distributions along the surfaces of the profiles. However, the use of such an approach requires solving a complete system of flow equations in the calculation domain, which limits its use when conducting a large number of parametric calculations and in optimal design problems. Further research, reported in [2], is aimed at improving the accuracy of the description of the flow in the flow parts of turbomachines by taking into account additional mechanisms for the formation of energy losses. It is shown that the characteristics of the stages are significantly affected by three-dimensional effects, secondary flows, and end gaps. However, taking into account these factors leads to an increase in the complexity of mathematical models and an increase in computational costs. For a more detailed description of the spatial structure of the flow, three-dimensional flow models based on the Navier-Stokes equations were used in [3]. This approach makes it possible to obtain detailed information about the velocity field, pressures, and loss parameters. At the same time, the need to discretize the entire volume of the flow part and take into account viscous effects significantly increases the dimensionality of the problem, which complicates the use of such models in tasks of repeated analysis and optimization.

In [4], the possibilities of using CFD approaches for calculating complex flows in turbomachines are considered. It is shown that increasing the physical reliability of the models is accompanied by an increase in the calculation time and requirements for computational resources. Thus, the issue of finding a compromise between the accuracy of the flow description and the efficiency of the calculation algorithm remains.

The task to increase the accuracy of mathematical modeling is considered in work [5], which analyzes current evolution of CFD methods for turbomachines. It is shown that the use of complex turbulence models and high spatial detail makes it possible to improve the prediction of flow parameters. However, such models have limited application in solving optimization problems because each change in geometry requires repeated execution of resource-intensive calculations.

In [6, 7], the possibilities of using high-precision approaches for modeling the structure of turbulent flows were investigated. In work [6], the use of a detailed spatial description of the flow made it possible to obtain information about the structure of vortex zones, flow mixing mechanisms, and the formation of aerodynamic losses. The advantage of such an approach is the possibility of direct analysis of complex physical phenomena that are difficult to take into account in simplified models. At the same time, the use of high spatial and temporal detail significantly increases the volume of calculations. This limits the application of such methods for tasks that require a significant number of calculation cycles, in particular in parametric studies and optimal design. In [7], the application of numerical modeling methods with increased accuracy for the analysis of gas-dynamic processes in turbomachines was considered. It was shown that taking into account the non-stationarity of the flow and detailed transport mechanisms makes it possible to significantly increase reliability in predicting the characteristics of the flow part. However, increasing the physical completeness of the mathematical model requires the use of complex calculation grids and significant computational resources. In addition, the direct application of such models in inverse design problems is complicated by the need for repeated calculations when changing geometric parameters. The results make it possible to analyze the mechanisms of loss formation and complex vortex structures. However, the use of such methods for real flow parts of turbomachines remains limited due to significant computational costs.

The task to reduce the computational cost of numerical modeling is considered in [8]. It is shown that even with the use of modern computational technologies, ensuring both high accuracy and acceptable speed remains challenging. In [9, 10], models were proposed for describing flows taking into account the real properties of the working fluid; however, increasing the level of physical description leads to a further increase in the complexity of calculations. In [9], approaches to numerical modeling of flows in turbomachines are considered, taking into account the actual properties of the working fluid and complex thermodynamic processes. The use of refined models makes it possible to increase accuracy in determining the flow parameters, especially with significant changes in temperature, pressure, and density. At the same time, taking into account additional physical effects leads to an increase in the number of calculation parameters and the complexity of the system of equations. This reduces the efficiency of such models when performing a large number of calculations necessary for optimizing structures.

In [10], numerical modeling of flows in the flow sections of turbomachines was investigated, taking into account variable thermodynamic properties of the gas and complex flow regimes. It was shown that such approaches provide a high correspondence between the calculated and experimental characteristics. However, increasing the accuracy of the model is accompanied by increasing the requirements for the calculation domain, grid detailing, and calculation execution time. As a result, the use of such methods for tasks of finding optimal geometry or solving inverse problems remains limited.

One of the reasons for the limited practical effectiveness of high-precision CFD approaches is the issue related to turbulence modeling. In [11], a two-parameter turbulence model was proposed, which became one of the most common in engineering practice due to the successful combination of accuracy and speed. At the same time, it was shown in [12] that there is no universal turbulence model and prediction errors increase significantly in flows with a complex structure, curvature of streamlines, and unfavorable pressure gradients. To overcome these difficulties, it was proposed in [13, 14] to use machine learning methods and high-precision modeling data to build new closure models.

Another promising direction was hybrid RANS–LES approaches whose evolution is described in [15]. In [13], the possibilities of using machine learning methods to build new closure models in CFD were considered. The potential for increasing the accuracy of predicting flow parameters was shown. However, large volumes of high-precision data are required to train such models. The authors of [14] proposed a multi-criteria approach to building turbulence models using CFD data. Accuracy was improved for certain classes of problems but the issue of universality of models remains unresolved. In [15], hybrid RANS–LES methods were investigated. The possibility of combining acceptable accuracy and relatively moderate computational costs was shown. At the same time, the complexity of implementing such approaches significantly exceeds the complexity of conventional engineering models.

In parallel with the development of CFD, aerodynamic optimization and inverse design methods were actively studied. In [16], the possibility of using control methods to determine the optimal geometry of aerodynamic surfaces was shown. Further advancement of this approach is reported in [17], in which a conjugate approach to determining the gradients of objective functions was devised. Its high efficiency compared to finite difference methods was shown. However, each optimization iteration still requires solving the direct flow problem.

The practical application of such approaches for turbomachines is considered in [18–20]. In [18], industrial optimization strategies for turbomachines are analyzed. The effectiveness of combining CFD and algorithms for finding optimal parameters is shown. In [19, 20], modern reviews of aerodynamic optimization methods for turbomachines are given. Significant progress in the application of conjugate methods and automated design is shown. It is the high computational cost that remains the main reason for the limited use of these methods in large-scale optimization problems. An alternative direction of development has been reduced-order methods and quasi-three-dimensional approaches. The theoretical foundations of such an approach were laid in [21], in which the general theory of three-dimensional flow in turbomachines was stated.

Further advancement was achieved through methods of quasi-three-dimensional analysis [22], which made it possible to take into account the spatial nature of the flow at significantly lower computational costs. In [23], a comparison of different end-to-end calculation methods was performed and their suitability for preliminary design was shown. The results from [24] confirmed the possibility of using such approaches for transonic turbine stages. To improve the accuracy of simplified models, a modern loss model was proposed in [25], which makes it possible to take into account the main mechanisms of energy dissipation without a significant increase in computational complexity. However, most existing quasi-three-dimensional methods focus mainly on solving direct problems and do not provide an effective solution to inverse design problems.

Thus, our review of existing approaches shows that current methods for calculating flows in turbomachines provide high accuracy in determining flow parameters but their application in operational design and optimization problems is limited by significant computational complexity. High-precision methods require solving large systems of equations taking into account viscous and turbulent effects, while reduced-order methods are mainly focused on solving direct problems and do not provide an effective statement of geometry recovery problems.

The development of a mathematical framework that would make it possible to simultaneously perform direct flow analysis and solve inverse design problems at a significant reduction in computational costs remains unsolved. It is especially important to ensure the possibility of local solution of the problem in cross-sections of the flow part without the need for multiple calculations of the full flow field. Overcoming these limitations is possible by using a special formulation of the flow equations, which preserves the basic laws of motion of the working medium and simultaneously reduces the dimensionality of the calculation problem. That predetermined the area of research into devising a computationally efficient method for solving direct and inverse aerodynamic problems in turbomachinery ducts based on the flow function.

## 3. The aim and objectives of the study

The aim of our study is to devise a computationally efficient method for solving direct and inverse aerodynamic problems in turbomachinery ducts based on the flow function, which enables reduction in computational costs while maintaining the accuracy of the description of the flow parameters and geometry of the flow parts of turbomachinery required for engineering analysis.

To achieve this goal, the following tasks were set:

– to define a single mathematical statement of direct and inverse aerodynamic problems for different types of turbomachinery ducts and to reduce flow equations to a compact system of differential and algebraic relations based on the flow function;

– to formulate direct and inverse aerodynamic problems as nonlinear optimization problems and to investigate the correctness of their mathematical statement;

– to verify the accuracy of the proposed method by comparing the results of calculations with experimental data and to assess the possibility of practical application of the method.

## 4. The study materials and methods

### 4. 1. The object and hypothesis of the study

The object of our study is aerodynamic processes in interblade, axial annular, and radial-axial channels of turbomachines. The study is aimed at determining the possibility of using a single mathematical statement based on the flow function to solve direct and inverse problems of aerodynamic design of elements in the flow parts of turbomachines.

The principal hypothesis assumes that the use of the flow function as the main calculation variable makes it possible to develop a single method for describing different types of flows in the channels of turbomachines and reduce the computational complexity of solving problems compared to full CFD models while maintaining the accuracy necessary for engineering analysis.

When building the mathematical model, it is assumed that the working medium is a compressible inviscid fluid. The flow is considered to be stationary, adiabatic, and continuous. Under these conditions, the total enthalpy, entropy, as well as isentrope index, remain constant along the streamlines, which makes it possible to use a simplified notation of the equations of motion.

The adopted simplifications are associated with the neglect of direct modeling of viscous effects, the development of the boundary layer, and complex turbulent structures. This assumption allows us to focus on determining the basic patterns of the velocity distribution and geometric parameters of the flow; the estimation of losses can subsequently be performed using empirical or semi-empirical relationships.

Our study considers a mathematical model of compressible flow in turbomachine channels [26-29]. The proposed method assumes three basic types of flow channels: interblade channels on surfaces of rotation, free section of an axial annular channel, and axisymmetric radial-axial annular channel. Under our assumptions, the basic equations of motion of a compressible fluid are transformed using a flow function that ensures that the flow continuity condition is met.

### 4. 2. Description of the mathematical model

The basic equations in the model:

– energy conservation equation (in absolute or relative statement)

$$i_0^* = i + \frac{C^2}{2}, \quad \frac{k}{k-1} pv + \frac{W^2}{2} - \frac{u^2}{2} = \text{const}; \tag{1}$$

– isentropic process equation

$$pv^k = \text{const}; \tag{2}$$

– continuity equation

$$\frac{\partial}{\partial R}\left(\frac{R}{v} C_R\right) + \frac{\partial}{\partial z}\left(\frac{R}{v} C_Z\right) = 0; \tag{3}$$

– the momentum equations projected onto the circumferential radial and axial directions:

$$C_r \frac{\partial C_u}{\partial r} + C_z \frac{\partial C_u}{\partial z} + \frac{C_u C_r}{r} = 0, \tag{4}$$

$$C_r \frac{\partial C_r}{\partial r} + C_z \frac{\partial C_r}{\partial z} - \frac{C_u^2}{r} = -v \frac{\partial p}{\partial r}, \tag{5}$$

$$C_r \frac{\partial C_z}{\partial r} + C_z \frac{\partial C_z}{\partial z} = -v \frac{\partial p}{\partial z}. \tag{6}$$

For inverse problems, the velocity distribution, and its derivatives along the streamline in the corresponding direction are specified as additional constraints (equations) [30]. For example, in a special case for the axial direction, the system takes the form $C = C(z)$ or $W = W(z)$, $\frac{dC(z)}{dz}$ or $\frac{dW(z)}{dz}$ or $\frac{d^2C(z)}{dz^2}$ or $\frac{d^2W(z)}{dz^2}$. For channels of other types, the velocity and its derivatives will depend only on the selected flow direction.

## 5. Results of developing and investigating a method for solving direct and inverse aerodynamic problems in turbomachinery ducts

### 5. 1. Results of defining the mathematical statement of flow problems in turbomachinery ducts

The general form of equations that meet the requirements for an arbitrary surface of rotation is defined in [29]. The work gives a single structure of equation reduction, which is acceptable for all three types of ducts [26–28, 30].

The key feature of the proposed method is solving equations based on the introduction of the flow function. The flow function $\Psi = m\bar{\Psi}$ was set as a normalized, twice continuously differentiated function whose range of change is [0, 1] (Fig. 1)

$$\bar{\Psi} = \bar{F} + x\bar{F} / 1 + x\bar{F}, \tag{7}$$

where "$x$" is some previously unknown continuous twice-differentiated function of actual variables, which is determined in the process of solving aerodynamic problems and can take both positive and negative values: $-1 <$ "$x$" $< \infty$.

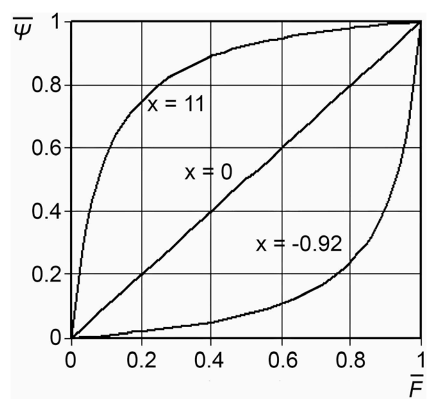



Fig. 1. Example of flow function $\bar{\Psi} = \bar{F} + x\bar{F} / 1 + x\bar{F}$ distributions

For interblade channels, the geometry is defined by the convex side (suction side) and the concave side (pressure side) (Fig. 2, *a*): $\varphi_s(l)$ and $\varphi_p(l)$ respectively, with the corresponding definition of the local cross-sectional area. The relative cross-sectional area of the channel is $\bar{F}(\theta, l) = \frac{\theta - \varphi_s(l)}{\varphi_p(l) - \varphi_s(l)}$, $0 \le \bar{F} \le 1$ (Fig. 2, *b*). For annular channels, the radii of the root (hub) and the periphery (tip) are set as $R_T(Z)$, which corresponds to the cross-sectional area (Fig. 2, *b*): $F(Z) = \pi\left(R_T^2 - R_H^2\right)$.

This method makes it possible to replace the continuity equation with an equivalent system that provides a consistent representation of the mass flow distribution in different types of channels, and thus the same mathematical structure is preserved for interblade and annular channels.

For the free section of the annular channel, the system of equations, which is equivalent to the continuity equation, is represented in the form

$$C_Z=\frac{V}{2\pi R}\frac{\partial G}{\partial R},\quad C_R=-\frac{V}{2\pi R}\frac{\partial G}{\partial Z},\quad \mathrm{tg}\gamma=-\frac{\partial G}{\partial Z}/\frac{\partial G}{\partial R}. \qquad (8)$$

For interblade channel:

$$W_l=\frac{v}{\tau r}\frac{\partial \Psi}{\partial \theta},\; W_u=-\frac{v}{\tau}\frac{\partial \Psi}{\partial l},$$

$$\frac{W_u}{W_l}=\mathrm{ctg}\beta=-\frac{\partial \Psi}{\partial l}/\left(\frac{1}{r}\frac{\partial \Psi}{\partial \theta}\right), \qquad (9)$$

For annular channel:

$$C_Z=\frac{V}{2\pi R}\frac{\partial G}{\partial R},\quad C_R=-\frac{V}{2\pi R}\frac{\partial G}{\partial Z},\quad \mathrm{tg}\gamma=-\frac{\partial G}{\partial Z}/\frac{\partial G}{\partial R}. \qquad (10)$$

At the next stage, the Euler equations are transformed, resulting in the dependences for determining the corresponding pressure gradient.

As a result, for the interblade channel, the following is obtained:

$$\frac{\partial p}{\partial \theta}=\frac{\left(M_{W_l}^2-1\right)\frac{v}{\tau^2}\frac{\partial \Psi}{\partial \theta}\left[\tau\frac{\partial}{\partial l}\left(\frac{1}{\tau}\frac{\partial \Psi}{\partial l}\right)+\frac{\mathrm{ctg}\beta}{r}\frac{\partial^2 \Psi}{\partial \theta \partial l}+\left(\frac{1}{r}\frac{\partial \Psi}{\partial l}-\frac{2\omega\tau}{v}\right)\sin\delta_z\right]}{M_{W_l}^2+M_{W_u}^2-1}-$$
$$-\frac{M_{W_l}^2\frac{v}{\tau^2}\left[\frac{\mathrm{ctg}\beta}{r}\frac{\partial^2 \Psi}{\partial \theta^2}+\tau r\frac{\partial}{\partial l}\left(\frac{1}{\tau r}\frac{\partial \Psi}{\partial \theta}\right)\right]}{M_{W_l}^2+M_{W_u}^2-1}+\frac{rA_l\sin\delta_z}{v}\frac{M_{W_l}M_{W_u}}{M_{W_l}^2+M_{W_u}^2-1}, \qquad (11)$$

$$\frac{\partial p}{\partial l}=\frac{v\frac{\partial \Psi}{\partial \theta}}{(r\tau)^2\left(M_{W_l}^2-1\right)}\times$$
$$\times\left[\begin{gathered}\frac{\mathrm{ctg}\beta}{r}\frac{\partial^2 \Psi}{\partial \theta^2}+\tau\frac{\partial}{\partial l}\left(\frac{1}{\tau}\frac{\partial \Psi}{\partial \theta}\right)-\\ -\frac{v}{a^2}\frac{\mathrm{ctg}\beta}{r}\frac{\partial \Psi}{\partial \theta}\frac{\partial p}{\partial \theta}\end{gathered}\right]-\frac{A_l\sin\beta}{\left(M_{W_l}^2-1\right)}, \qquad (12)$$

where $M_{W_l}$ and $M_{W_u}$ are the Mach numbers determined from the flow velocity projections $W_l$ and $W_u$.

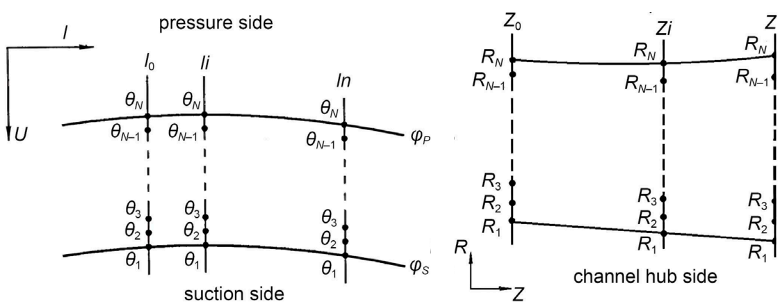



Fig. 2. Designations for the boundaries of sections in turbomachine channels: *a* — interblade channel; *b* — annular channel

In an axial-radial annular channel, in the general case

$$\frac{\partial p}{\partial m}=f\left(m,p,C_u,\frac{\partial \Psi}{\partial m},\frac{\partial \Psi}{\partial n},\frac{\partial^2 \Psi}{\partial m^2},\frac{\partial^2 \Psi}{\partial n^2},\frac{\partial^2 \Psi}{\partial m \partial n}\right). \qquad (13)$$

In the special case when the angle of rotation of the axes (Fig. 3) is $\alpha = 0$

$$\frac{\partial p}{\partial R}=\frac{1-M_{C_Z}^2}{1-M_{C_Z}^2-M_{C_R}^2}\frac{v}{(2\pi R)^2}\times$$
$$\times\frac{\partial G}{\partial R}\left[\frac{M_{C_Z}M_{C_R}}{1-M_{C_Z}^2}B_1-B_2\right]+\frac{C_U^2}{vR}, \qquad (14)$$

where $M_{C_Z}=C_Z/a$, $M_{C_R}=C_R/a$ – Mach numbers determined from the axial and radial components of the flow velocity; $a=\sqrt{kpv}$ – speed of sound:

$$B_1=\left(\frac{1}{R}\frac{\partial G}{\partial R}-\frac{\partial^2 G}{\partial R^2}\right)\mathrm{tg}\gamma-\frac{\partial^2 G}{\partial R \partial Z},$$

$$B_2=\left(\frac{1}{R}\frac{\partial G}{\partial Z}-\frac{\partial^2 G}{\partial Z \partial R}\right)\mathrm{tg}\gamma-\frac{\partial^2 G}{\partial Z^2}.$$

For the free section of the axial annular channel

$$\frac{\partial P}{\partial R}=\frac{1-M_{C_Z}^2}{1-M_{C_Z}^2-M_{C_R}^2}\left\{\frac{M_{C_Z}M_{C_R}}{1-M_{C_Z}^2}VB_5B_4-B_5B_2+\frac{C_U^2}{VR}\right\}, \qquad (15)$$

where

$$B_5B_3=\frac{1}{(2\pi R)^2}\frac{\partial G}{\partial R}\frac{V^2}{a^2}\frac{\partial G}{\partial R}=$$
$$=\frac{V^2}{(2\pi R)^2}\left(\frac{\partial G}{\partial R}\right)^2\frac{1}{a^2}=\frac{C_Z^2}{a^2}=M_{C_Z}^2,$$

$$B_5B_1=\frac{1}{(2\pi R)^2}\frac{\partial G}{\partial R}\frac{\partial G}{\partial Z}\frac{V^2}{a^2}=$$
$$=\frac{V}{2\pi R}\frac{\partial G}{\partial R}\frac{V}{2\pi R}\frac{\partial G}{\partial Z}\frac{1}{a^2}=$$
$$=-\frac{C_ZC_R}{a^2}=-M_{C_Z}M_{C_R},$$

$$B_5B_1\mathrm{tg}\gamma=-\frac{C_Z\mathrm{tg}\gamma C_R}{a^2}=-\frac{C_RC_R}{a^2}=-M_{C_R}^2,$$

$$B_5B_2=\frac{V}{(2\pi R)^2}\frac{\partial G}{\partial R}\times$$
$$\times\left[\left(\frac{1}{R}\frac{\partial G}{\partial Z}-\frac{\partial^2 G}{\partial Z \partial R}\right)\mathrm{tg}\gamma-\frac{\partial^2 G}{\partial Z^2}\right],$$

$$B_5B_2=\frac{V}{(2\pi R)^2}\frac{\partial G}{\partial R}\times$$
$$\times\left[\left(\frac{1}{R}\frac{\partial G}{\partial Z}-\frac{\partial^2 G}{\partial Z \partial R}\right)\mathrm{tg}\gamma-\frac{\partial^2 G}{\partial Z^2}\right].$$

Thus, the Euler equation was transformed, which allowed us to obtain expressions for pressure gradients in the form of functions of the derivatives of the flow function, velocity components, and Mach numbers.

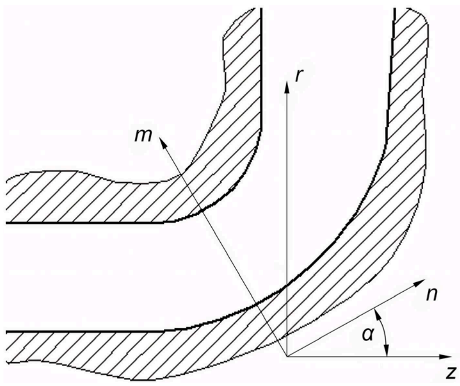



Fig. 3. Coordinate directions in the meridional plane

**5. 2. Establishing procedures for solving direct and inverse problems**

This subchapter considers an example of an axisymmetric flow in annular channels, for which the solution was obtained in the meridional plane. The transformation of coordinates from $r$ and $z$ into orthogonal coordinates $m$ and $n$ (Fig. 3), rotated by an angle $\alpha$, is introduced.

The application of such a transformation allowed us to construct an arbitrary family of lines $n$ = const, which determined the calculated cross-sections of the channel. The origin of the coordinate directions $m$ and $n$ was located on the $z$ axis, while the possibility of its displacement in the radial direction was provided:

$$r = n \cdot \sin\alpha + m \cdot \cos\alpha, \;\; C_Z = C_n \cdot \cos\alpha - C_m \cdot \sin\alpha,$$

$$Z = n \cdot \cos\alpha - m \cdot \sin\alpha, \;\; C_r = C_n \cdot \sin\alpha + C_m \cdot \cos\alpha,$$

$$\frac{\partial C_r}{\partial Z} = -\frac{\partial C_r}{\partial m}\sin\alpha + \frac{\partial C_r}{\partial n}\cos\alpha,$$

$$\frac{\partial C_Z}{\partial Z} = -\frac{\partial C_Z}{\partial m}\sin\alpha + \frac{\partial C_Z}{\partial n}\cos\alpha,$$

$$\frac{\partial C_r}{\partial r} = \frac{\partial C_r}{\partial m}\cos\alpha + \frac{\partial C_r}{\partial n}\sin\alpha,$$

$$\frac{\partial C_Z}{\partial r} = \frac{\partial C_Z}{\partial m}\cos\alpha + \frac{\partial C_Z}{\partial n}\sin\alpha. \quad (16)$$

We set coordinates for the inner boundary $m_k = m_k(n_i)$ and the outer boundary of the channel $m_p = m_p(n_i)$, and their derivatives $\frac{dm_k(n_i)}{dn}, \frac{dm_p(n_i)}{dn}, \frac{d^2m_k(n_i)}{dn^2}, \frac{d^2m_p(n_i)}{dn^2}$. The solution to the flow calculation problem is considered complete when the variables $x$ of the flow function (7) are defined along the lines $n = n_i$ or in a reasonably dense set of cross sections $n$ = const. The problem for the interblade channel can be represented similarly. The problem of calculating the flow in a certain cross section $l$ = const (Fig. 2, *a*) is defined as a partial problem, while its independence from other partial problems is established.

The solution to the problem of determining the flow parameters in a predefined cross-section $l$ = const was investigated. The principle is similar to that for other types of channels. In the cross-section $n = n_i$, the required number of equidistant points $m_j$, $j = \overline{1,N}$, $m_k = m_k(n_i)$, $m_p = m_p(n_i)$ was set. The values of the independent variables "$x$" of the flow function are set, and, at the points, we calculate the values of the dimensionless flow function (7), its partial derivatives up to the second order inclusive, and the circumferential components of the flow velocity (10), (16).

At point $m_j$ with numbers $j$ = 1 or $j = N$, the $p_1$ or $p_N$ pressure was determined, which satisfied the energy conservation equation

$$\frac{2k}{k-1}\left(p_j^* v_j^* 0 p_j v_j^* \left(\frac{p_j^*}{p_j}\right)^{\frac{1}{k}}\right) =$$
$$= \left[\frac{G v_j^*}{2\pi m_j (\cos\alpha + n_i \cos\alpha)}\left(\frac{p_j^*}{p_j}\right)^{\frac{1}{k}}\right] \times$$
$$\times \left[\left(\frac{\partial \bar{\Psi}}{\partial m}\right)_j^2 + \left(\frac{\partial \bar{\Psi}}{\partial n}\right)_j^2\right] + C_{uj}^{2\,2}. \quad (17)$$

The solution to the equation for pressure gradient $\frac{dp}{dm} = f(m,p)$ in the cross section $n = n_i$ was defined as the solution to the Cauchy problem on the interval $[m_k(n_i), m_p(n_i)]$ under the boundary condition set by pressure value $p_1$ or pressure $p_N$.

For arbitrarily defined actual variables of the flow function (for example, zero values), pressure values were determined at all points of the channel cross section. Condition (17) was met only at one point: at point $m_1 = m_k(n_i)$, or $m_N = m_p(n_i)$. Therefore, the fulfillment of the energy conservation equation depended on the correct choice of actual variables "$x$" (7). The specific problem was stated as a nonlinear programming problem, in which the objective function was defined as a quantitative estimate of the fulfillment of equation (17) at points $m_j$, $j = \overline{1,N}$. The mass flow rate through the channel, calculated from equation (17), and the mass flow rate predefined by the conditions of the specific problem should not differ at the points set.

Therefore, the solution was derived independently for each cross-section, and the cross-section itself was discretized into a set of points. At these points, the flow function and its derivatives were calculated, the velocity components were set (inverse problem) or determined (direct problem); the boundary pressure was determined from the energy equation, and the pressure field inside the cross-section was obtained by solving the Cauchy problem.

The initial data for the calculation: mass flow rate, total pressure and specific volume, geometry of the channel boundaries and their derivatives. For the inverse problem, the geometry of the streamline and the velocity distribution along it.

The result of solving the direct problem is the reconstruction of the full set of flow variables within the cross-section, and for the inverse problem, determining the geometry of channel boundaries. The solution depends on the unknown parameters – "$x$" function, which determine the nature of change in the flow function.

Thus, each local problem is stated as a nonlinear optimization problem where the design variables "$x$" are the parameters of the flow function, the objective function is the sum of the squares of the mass flow rate residuals estimated from the energy equation.

The proposed flow function-based statement reduces the complete system of governing equations to a compact and computationally efficient form.

As a result, the calculation of the flow in each cross-section is transformed into an independent nonlinear optimization problem, which significantly simplifies the solution procedure while maintaining physical consistency.

**5. 3. Verifying the accuracy of solving aerodynamic and problems in the ducts of turbomachines**

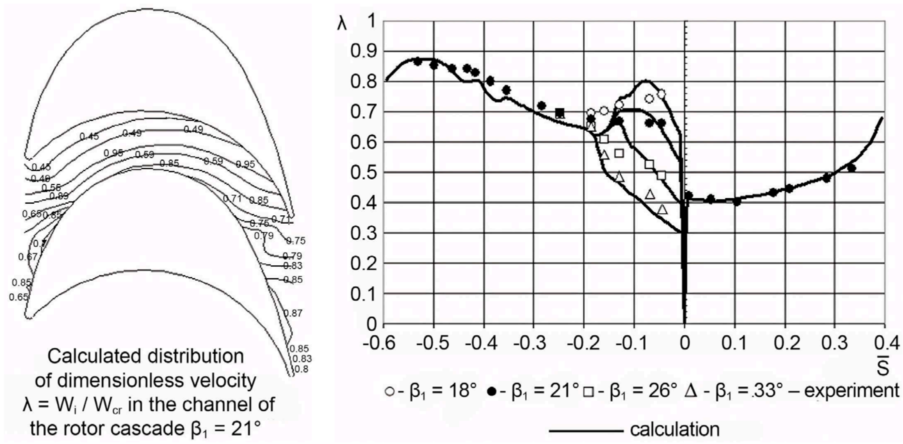
**5. 3. 1. Investigating the accuracy of solving direct aerodynamic and problems in the ducts of turbomachines**

Fig. 4–6 show a comparison of experimental [30] and calculated data in the interblade ducts of two rotor cascades of turbines TP-1A (P-2617A) and TP-2A (P-3021A), and the nozzle cascade TC-1A (C-9012A, H-4) [31, 32].

For the TP-1A cascade, the calculation results are represented as follows: the distribution of the dimensionless velocity in the interblade channel at the angle of the inlet flow without shocks $\beta_1 = 21°$ (Fig. 4), and the distribution of the dimensionless velocity along the profile surface at different angles of the inlet flow $\beta_1 = 18°, 21°, 26°, 33°$ (Fig. 4).

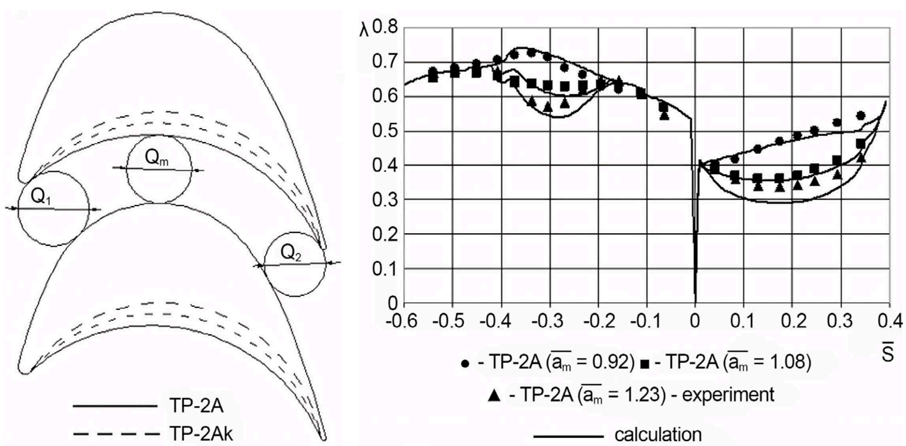
Calculations for the TP-2A type cascades were carried out in order to determine the distribution of flow parameters in the diffuser-confusion channels of cascades. In the original TP-2A cascade, the curvature of the concave surface of the profile was increased by cutting [30], while the convex surface remained unchanged. As a result, the interblade channel first expanded and then narrowed. The basic geometric parameters: $a_1$ – width of the inlet opening, $a_m$ – width of the middle of the channel, $a_2$ – width of the outlet throat (Fig. 5). The original TP-2A cascade is characterized by the following geometric ratios $\bar{a}_m = a_m / a_1 = 0.92$ and $\bar{a}_1 = a_1 / a_2 = 1.16$, while the two modified TP-2Ak cascades have diffusion ratios $\bar{a}_m = 1.08$ and 1.23 at the same $\bar{a}_1 = a_1 / a_2 = 1.16$.

Basic geometric characteristics of the TC-1A nozzle cascade are shown in Fig. 6. Experimental studies on this cascade are reported in [31, 32] under the following conditions: Mach number 0.4, Reynolds number $7 \cdot 10^5$. Comparison with experimental data is carried out using the distribution of dimensionless velocity $\bar{C} = C_i / C_0$ ($C_i$ – current value of velocity on the profile surface, $C_0$ – flow velocity at the inlet to the cascade) along the profile surface where the local velocity is related to the inlet velocity (Fig. 6).



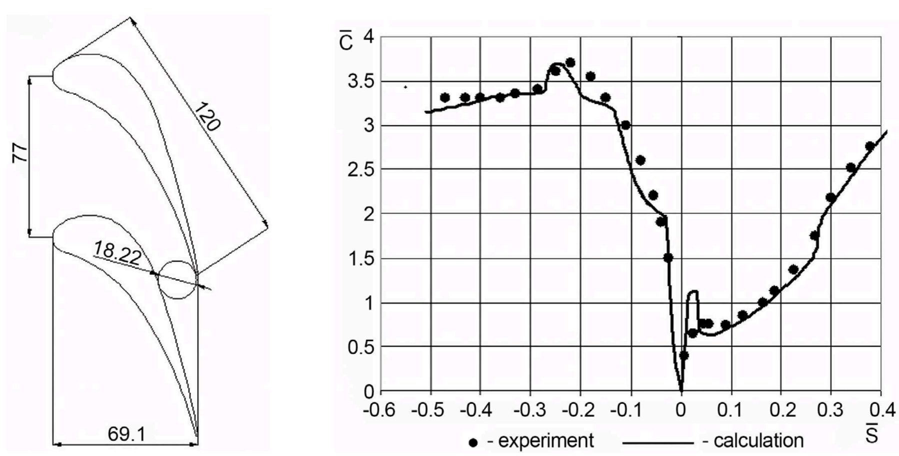

*a* *b*

Fig. 4. Distribution of dimensionless velocity $\lambda$ in the TP-1A profile cascade at different flow inlet angles: *a* — $\lambda$ in the interblade channel; *b* — $\lambda$ along the profile contours




*a* *b*

Fig. 5. Distribution of dimensionless velocity $\lambda$ along the contour of profiles of rotor cascade TR-2A and TR-2Ak: *a* — geometric ratios of the rotor cascade s; *b* — $\lambda$ along the contours of profile




*a* *b*

Fig. 6. Distribution of dimensionless velocity $\bar{C}$ along the contour of the nozzle cascade profile TC-1A: *a* — geometric ratios of the nozzle cascade; *b* — $\bar{C}$ along the contours of profile

Thus, the devised method for solving a direct problem makes it possible, at satisfactory accuracy, to perform calculations in the confuser, diffuser-confuser, and diffusion channels of turbomachinery cascades.

To assess reliability of the calculated flow characteristics in the cross section of the free annular channel, a comparison was performed to experimental data [33, 34].

An axial turbine stage was investigated (Fig. 7) [33]. Flow parameters were measured in the end cross sections of free annular gaps after the rows of blades using gas-dynamic probes.

Our comparison of the calculated and experimental results is illustrated in Fig. 8–10, which show the distributions of static pressure $P$ and two velocity components $\bar{C}_{1z}$ and $\bar{C}_{1u}$ along cross section 1 (section 1) (velocities are referred to velocity $C_0$, equivalent to the stage difference). Similar distributions of mass flow, static pressure $P_2$, velocity and its components $\bar{C}_{2z}$ and $\bar{C}_{2u}$ along the blade height were obtained both behind the stator in cross section 2 (section 2) and behind the rotor in cross section 2 (section 2) (Fig. 11–13).

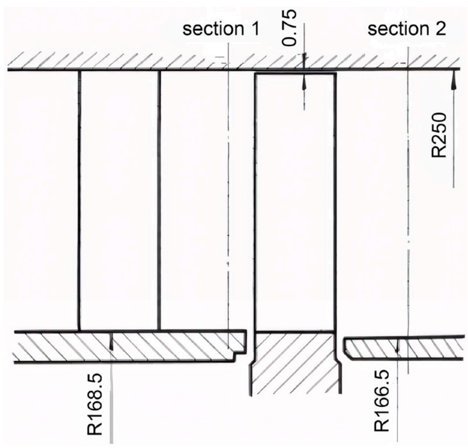



Fig. 7. Basic geometric characteristics of the turbine stage

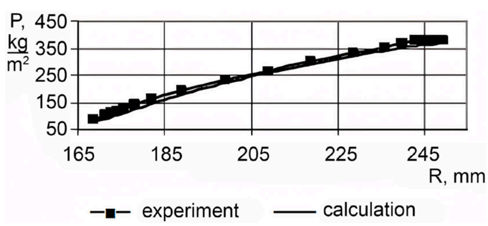



Fig. 8. Static pressure $P$ distribution along cross section 1

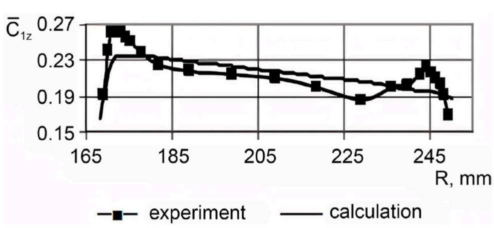



Fig. 9. Distribution of velocity component $\bar{C}_{1z}$ along cross section 1

The comparison confirmed that the devised method for free annular channels enabled determining the radial distribution of flow parameters after turbine cascades at acceptable accuracy. Our results indicate the possibility of using the proposed method for the analysis and design of multi-stage sections, as well as for the preliminary formation of flow parameter distributions when designing transition and outlet channels in turbomachines.

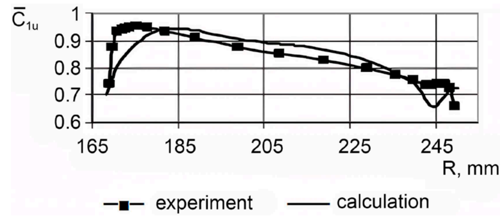



Fig. 10. Distribution of velocity component $\bar{C}_{1u}$ along cross section 1

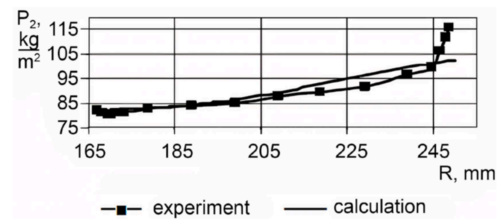



Fig. 11. Pressure $P_2$ distribution along cross section 2

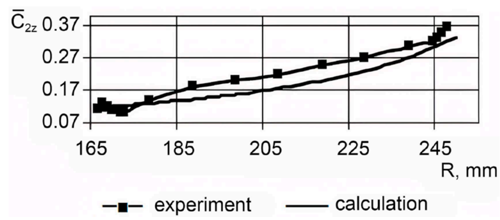



Fig. 12. Distribution of velocity component $\bar{C}_{2z}$ along cross section 2

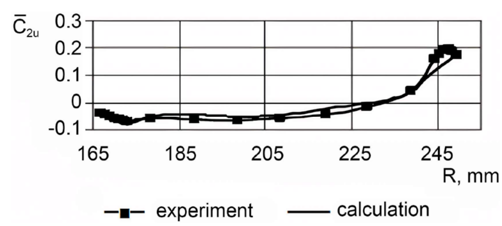



Fig. 13. Distribution of velocity component $\bar{C}_{2u}$ along cross section 2

**5. 3. 2. Investigating the accuracy of solving inverse aerodynamic problems**

A transonic turbine rotor cascade in an aircraft turbojet engine was chosen as a test case [30]. From the direct solution to the problem on a cylindrical flow surface, the following was obtained: the distribution of flow parameters in the interblade channel, the profile loss coefficient based on the velocity distribution, the geometry of the mean streamline, and the velocity distribution along it (used as boundary conditions for the inverse problem).

Using these boundary conditions, the inverse problem was solved. The results of the comparison are shown in Fig. 14.

It should be noted that the accuracy of determining the profile boundaries in the inverse problem depends both on the accuracy of the boundary conditions and the accuracy of the solution itself. They, in turn, depend on the accuracy of the statement and solution to the direct problem. For the devised methods, the difference between the coordinates of the original profile and the profile reconstructed using the inverse problem does not exceed 0.03 mm. Thus, the solutions to the direct and inverse problems demonstrate comparable accuracy. The velocity distribution along the surfaces of the original and reconstructed profiles is shown in Fig. 15.

The direct problem was solved for a diffuser with a length of $l = 500$ mm and an opening angle of $\alpha = 37.5°$. In total, 501 design cross sections $l =$ const were considered [33].

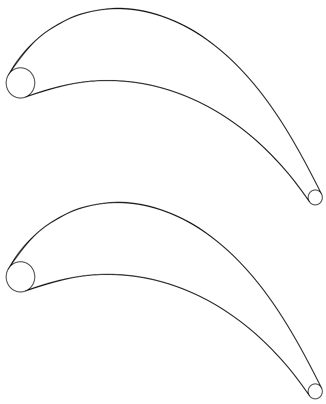


Fig. 14. The initial profile and the profile designed using the inverse problem solution

As a result, the flow parameters in each cross section were obtained, as well as the geometry of the average streamline and the velocity distribution along it. Fig. 16, 17 show a comparison between the initial geometry of the diffuser and the geometry obtained from the inverse problem, as well as the corresponding boundary conditions. The boundaries of the diffuser obtained from the inverse problem are marked with triangles in Fig. 16. Table 1 gives relative deviations in the radial coordinates for the surfaces of the inner $\Delta R_{hub}$ and outer $\Delta R_{tip}$ contours at 51 cross sections $l =$ const.

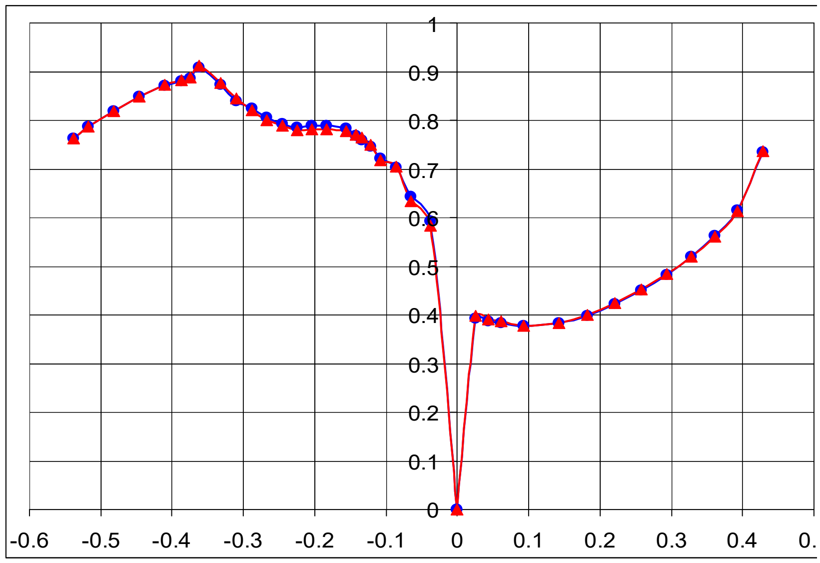



Fig. 15. Distribution of dimensionless flow velocity $\lambda = W_i/W_{kr}$ along the contours of profiles on a cylindrical surface of revolution: direct problem; inverse problem

Table 1

Relative deviations in the radial coordinate for the sleeve and the shell in cross sections $l =$ const

| Hub | | | | | | Tip | | | |
|---|---|---|---|---|---|---|---|---|---|
| Section No. | Coordinate $l$, m | $\Delta R_{hub}$, % | Section No. | Coordinate $l$, m | $\Delta R_{hub}$, % | Section No. | $\Delta R_{tip}$, % | Section No. | $\Delta R_{tip}$, % |
| 1 | 0 | –0.013 | 26 | 0.25 | –0.017 | 1 | 0.105 | 26 | 0.097 |
| 2 | 0.01 | –0.012 | 27 | 0.26 | –0.017 | 2 | 0.102 | 27 | 0.097 |
| 3 | 0.02 | –0.011 | 28 | 0.27 | –0.016 | 3 | 0.100 | 28 | 0.097 |
| 4 | 0.03 | –0.010 | 29 | 0.28 | –0.016 | 4 | 0.098 | 29 | 0.096 |
| 5 | 0.04 | –0.010 | 30 | 0.29 | –0.015 | 5 | 0.097 | 30 | 0.096 |
| 6 | 0.05 | –0.009 | 31 | 0.3 | –0.015 | 6 | 0.096 | 31 | 0.096 |
| 7 | 0.06 | –0.009 | 32 | 0.31 | –0.014 | 7 | 0.095 | 32 | 0.096 |
| 8 | 0.07 | –0.009 | 33 | 0.32 | –0.013 | 8 | 0.095 | 33 | 0.096 |
| 9 | 0.08 | –0.009 | 34 | 0.33 | –0.013 | 9 | 0.095 | 34 | 0.096 |
| 10 | 0.09 | –0.009 | 35 | 0.34 | –0.012 | 10 | 0.095 | 35 | 0.097 |
| 11 | 0.1 | –0.009 | 36 | 0.35 | –0.012 | 11 | 0.095 | 36 | 0.097 |
| 12 | 0.11 | –0.009 | 37 | 0.36 | –0.013 | 12 | 0.096 | 37 | 0.098 |
| 13 | 0.12 | –0.010 | 38 | 0.37 | –0.014 | 13 | 0.096 | 38 | 0.098 |
| 14 | 0.13 | –0.010 | 39 | 0.38 | –0.016 | 14 | 0.096 | 39 | 0.099 |
| 15 | 0.14 | –0.011 | 40 | 0.39 | –0.020 | 15 | 0.097 | 40 | 0.100 |
| 16 | 0.15 | –0.011 | 41 | 0.4 | –0.025 | 16 | 0.097 | 41 | 0.100 |
| 17 | 0.16 | –0.012 | 42 | 0.41 | –0.033 | 17 | 0.098 | 42 | 0.100 |
| 18 | 0.17 | –0.013 | 43 | 0.42 | –0.043 | 18 | 0.098 | 43 | 0.100 |
| 19 | 0.18 | –0.013 | 44 | 0.43 | –0.056 | 19 | 0.098 | 44 | 0.099 |
| 20 | 0.19 | –0.014 | 45 | 0.44 | –0.074 | 20 | 0.098 | 45 | 0.098 |
| 21 | 0.2 | –0.015 | 46 | 0.45 | –0.096 | 21 | 0.098 | 46 | 0.095 |
| 22 | 0.21 | –0.015 | 47 | 0.46 | –0.124 | 22 | 0.098 | 47 | 0.091 |
| 23 | 0.22 | –0.016 | 48 | 0.47 | –0.158 | 23 | 0.098 | 48 | 0.086 |
| 24 | 0.23 | –0.016 | 49 | 0.48 | –0.200 | 24 | 0.098 | 49 | 0.079 |
| 25 | 0.24 | –0.017 | 50 | 0.49 | –0.251 | 25 | 0.097 | 50 | 0.069 |
| – | – | – | 51 | 0.5 | –0.313 | – | – | 51 | 0.057 |

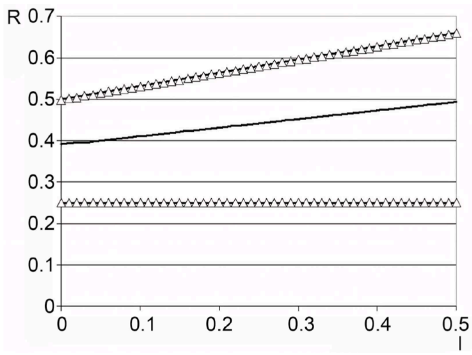



Fig. 16. Comparison of the geometry of the initial diffuser and the results of solving the inverse problem for the diffuser $l = 0.5$ m

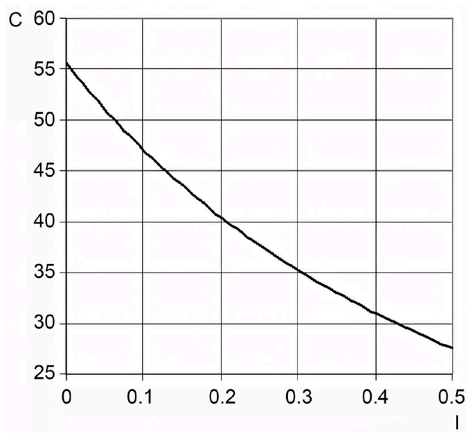



Fig. 17. Flow velocity distribution along the length of the diffuser

Analysis of the results given in Table 1 reveals that the geometry of the diffuser channel restored using the inverse problem has a high correspondence to the original configuration. The maximum relative deviations in the radial coordinates for the inner and outer contours do not exceed the established limits, which indicates sufficient accuracy in determining the boundaries of the flow channel for the predefined flow parameters.

Our results confirm that the proposed inverse solution procedure provides consistency between the predefined aerodynamic characteristics and the resulting channel geometry. This demonstrates the possibility of using the method for problems of preliminary design of flow parts in turbomachines when it is necessary to quickly assess the impact of changing geometric parameters on the flow structure.

## 6. Discussion of results based on investigating a method for solving direct and inverse aerodynamic problems in turbomachine channels

Our results are attributed to the features of the mathematical statement, in which the flow function is used as the main variable, and the execution of the energy conservation equation is controlled through the mass flow rate matching condition formulated on the basis of equation (17). The use of the flow function parameters *x* defined in formula (7) allowed us to reduce the problem of calculating the flow in each cross section to an independent nonlinear optimization problem. It is this structure of the statement that provided the possibility to locally determine flow parameters without the need to solve the global system of equations.

The reliability of results based on the direct problem is confirmed by comparing the calculated and experimental data shown in Fig. 4–6. The matching of velocity distributions indicates that the proposed statement correctly reproduces the main regularities of compressible flow in the interblade channels of turbomachines. The result is explained by the fact that the flow function automatically ensures that the continuity equation is fulfilled while the use of Euler equations makes it possible to correctly take into account the influence of channel geometry on the flow structure.

The high accuracy of the inverse problem solution is confirmed by the results shown in Fig. 15–17 and given in Table 1. A comparison of velocity distributions in Fig. 15 illustrates a practical coincidence of the results from the direct and inverse problems; the data in Fig. 16, 17 and Table 1 indicate high accuracy in the reconstruction of the channel geometry. The maximum deviations in the blade profiles did not exceed 0.03 mm, and they were less than 0.32% for the diffuser. This result is explained by the fact that the optimization objective function is directly related to the fulfillment of the mass flow conservation condition, therefore the geometry is reconstructed in accordance with the specified aerodynamic characteristics.

Compared with the known CFD approaches based on solving the Navier-Stokes equations and RANS-, LES-, or DNS-modeling [6–10], the proposed method is characterized by significantly lower computational complexity. Unlike conventional inverse methods that require multiple solutions to direct problems or the use of adjoint equations [16–20], the inverse problem in our method is stated locally in each cross-section. This makes it possible to significantly reduce the number of unknowns and reduce computational costs without losing engineeringly acceptable accuracy. At the same time, unlike classical end-to-end methods [21–24], the proposed statement makes it possible to solve direct and inverse problems within a single mathematical model.

At the same time, a certain limitation of the proposed method is the use of an inviscid flow formulation based on Euler equations, which does not provide direct modeling of friction processes, boundary layer development, flow detachment, and secondary flows. However, determining losses in the flow parts of turbomachines does not necessarily require a complete solution to the viscous problem because there are empirical and semi-empirical dependences that make it possible to estimate losses based on flow parameters, in particular, on the distribution of velocity, pressure, and geometric characteristics of channels. The use of such models in combination with the proposed method could ensure that the main loss mechanisms are taken into account without a significant increase in computational complexity.

Further studies may involve the integration of loss models and simplified turbulence models into the constructed system of equations, as well as the expansion of the algorithm for flows with pronounced viscous effects. The combination of the proposed method with semi-empirical loss estimation models when solving problems of optimal design of flow parts of turbomachines is especially promising. Such integration would make it possible to use the obtained velocity distributions and flow parameters not only to determine the geometry of the channels but also for a comprehensive assessment of their aerodynamic efficiency when searching for optimal design solutions. The main difficulties related to such advancement might involve ensuring the consistency of semi-empirical dependences with the mathematical notation of the method, maintaining computational efficiency and stability in solving inverse problems at an increase in the number of design parameters. At the same time, our results allow us to argue about the prospects of using the proposed method for preliminary design, parametric studies, and optimization of elements in the flow parts of turbomachines.

## 7. Conclusions

1. A unified mathematical representation of direct and inverse aerodynamic problems for various types of turbomachinery ducts has been devised. The use of the flow function as the main variable has allowed us to reduce the compressible flow equations to a compact system of differential and algebraic relations, which provided the possibility of describing interblade, axial annular, and radial-axial ducts within the framework of a single approach. That made it possible to decompose the initial multidimensional problem into local subproblems in the duct cross-sections, due to which the computational complexity was reduced by at least an order of magnitude compared to conventional CFD approaches. The resulting effect is explained by the reduction in the number of interconnected unknowns and the absence of the need to solve large global systems of equations.

2. A procedure for solving direct and inverse problems based on local decomposition of the computational domain and statement of the problem of determining the flow parameters as a nonlinear optimization problem has been proposed. This method allowed us to reduce dimensionality of the problem and ensure the restoration of the geometry of flow channels with a deviation of no more than 0.03 mm for the blade profiles and 0.32% for the diffuser channel. This indicates the capability of the proposed method to provide the necessary accuracy for inverse design problems.

3. The proposed method was verified by comparing the calculated results with experimental data and the results of direct calculations for turbomachinery channels. Consistent velocity and pressure distributions in the interblade, annular, and diffuser channels were obtained, while the discrepancy between the calculated and reference geometric parameters did not exceed 0.3%. Our results confirm the possibility of using the proposed method for preliminary design, parametric studies, and optimization problems of flow parts in turbomachines.

## Conflicts of interest

The authors declare that they have no conflicts of interest in relation to the current study, including financial, personal, authorship, or any other, that could affect the study and the results reported in this paper.

## Funding

The study was conducted without financial support.

## Data availability

The datasets used and analyzed during this study are available from the corresponding author upon reasonable request.

## Use of artificial intelligence

The authors declare the use of generative AI in the research and manuscript preparation process. According to the GAIDeT taxonomy (2025), the following tasks were delegated to generative AI tools under full human supervision:

– literature search (all references to literature sources were checked by the authors).

Generative AI tool used: Chart GPT-5.5.

The authors bear full responsibility for the final manuscript.

## Authors' contributions

**Valery Subotovich:** Conceptualization, Methodology; **Oleksandr Yudin:** Methodology, Software, Validation; Writing – review & editing**; Artem Babaiev:** Software, Data Curation, Writing – review & editing; **Valentyn Barannik:** Validation, Data Curation; **Olena Avdieieva:** Writing – original draft, Writing – review & editing.

References


1. Denton, J. D. (1983). An Improved Time-Marching Method for Turbomachinery Flow Calculation. Journal of Engineering for Power, 105 (3), 514–521. https://doi.org/10.1115/1.3227444
2. Denton, J. D. (1993). The 1993 IGTI Scholar Lecture: Loss Mechanisms in Turbomachines. Journal of Turbomachinery, 115 (4), 621–656. https://doi.org/10.1115/1.2929299
3. Dawes, W. N. (1993). The Extension of a Solution-Adaptive Three-Dimensional Navier–Stokes Solver Toward Geometries of Arbitrary Complexity. Journal of Turbomachinery, 115 (2), 283–295. https://doi.org/10.1115/1.2929234
4. Denton, J. D., Dawes, W. N. (1998). Computational fluid dynamics for turbomachinery design. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 213 (2), 107–124. https://doi.org/10.1243/0954406991522211
5. Dawes, W. N. (2007). Turbomachinery computational fluid dynamics: asymptotes and paradigm shifts. Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences, 365 (1859), 2553–2585. https://doi.org/10.1098/rsta.2007.2021
6. Sandberg, R. D., Michelassi, V. (2019). The Current State of High-Fidelity Simulations for Main Gas Path Turbomachinery Components and Their Industrial Impact. Flow, Turbulence and Combustion, 102 (4), 797–848. https://doi.org/10.1007/s10494-019-00013-3
7. Sandberg, R. D., Michelassi, V. (2022). Fluid Dynamics of Axial Turbomachinery: Blade- and Stage-Level Simulations and Models. Annual Review of Fluid Mechanics, 54 (1), 255–285. https://doi.org/10.1146/annurev-fluid-031221-105530

8. Slotnick, J., Khodadoust, A., Alonso, J., Darmofal, D., Gropp, W., Lurie, E., Mavriplis, D. J (2014). CFD Vision 2030 Study: A Path to Revolutionary Computational Aerosciences. NASA Technical Report. Available at: https://ntrs.nasa.gov/citations/20140003093
9. Wheeler, A. P. S. (2024). High fidelity simulation of dense vapours with thermodynamic consistent modelling. Computers & Fluids, 268, 106088. https://doi.org/10.1016/j.compfluid.2023.106088
10. Zhang, E., Watanabe, T., Lai, Z., Bai, B. (2024). A compressible flow solver for turbomachinery of the real gases with strongly variable properties. Energy, 290, 129915. https://doi.org/10.1016/j.energy.2023.129915
11. Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal, 32 (8), 1598–1605. https://doi.org/10.2514/3.12149
12. Pope, S. B. (2000). Turbulent Flows. Cambridge University Press. https://doi.org/10.1017/cbo9780511840531
13. Hammond, J., Pepper, N., Montomoli, F., Michelassi, V. (2022). Machine Learning Methods in CFD for Turbomachinery: A Review. International Journal of Turbomachinery, Propulsion and Power, 7 (2), 16. https://doi.org/10.3390/ijtpp7020016
14. Waschkowski, F., Zhao, Y., Sandberg, R., Klewicki, J. (2022). Multi-objective CFD-driven development of coupled turbulence closure models. Journal of Computational Physics, 452, 110922. https://doi.org/10.1016/j.jcp.2021.110922
15. Spalart, P. R. (2021). Hybrid RANS-LES Methods. Advanced Approaches in Turbulence, 133–159. https://doi.org/10.1016/b978-0-12-820774-1.00010-0
16. Jameson, A. (1988). Aerodynamic design via control theory. Journal of Scientific Computing, 3 (3), 233–260. https://doi.org/10.1007/bf01061285
17. Giles, M. B., Pierce, N. A. (2000). An Introduction to the Adjoint Approach to Design. Flow, Turbulence and Combustion, 65 (3-4), 393–415. https://doi.org/10.1023/a:1011430410075
18. Shahpar, S. (2010). Optimisation Strategies Used in Turbomachinery Design from an Industrial Perspective. In book: INTRODUCTION TO OPTIMIZATION AND MULTIDISCIPLINARY DESIGN IN AERONAUTICS AND TURBOMACHINERY. Vol. 1. Chap. 7. Publisher: Von-Karmaan Institute (VKI). Available at: https://www.researchgate.net/publication/267763601_Optimisation_strategies_used_in_turbomachinery_design_from_an_industrial_perspective
19. Li, Z., Zheng, X. (2017). Review of design optimization methods for turbomachinery aerodynamics. Progress in Aerospace Sciences, 93, 1–23. https://doi.org/10.1016/j.paerosci.2017.05.003
20. Lavimi, R., Benchikh Le Hocine, A. E., Poncet, S., Marcos, B., Panneton, R. (2024). A review on aerodynamic optimization of turbomachinery using adjoint method. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 238 (13), 6405–6441. https://doi.org/10.1177/09544062231221625
21. Wu, C.-H. (1952). A General Theory of Three-Dimensional Flow in Subsonic and Supersonic Turbomachines of Axial, Radial, and Mixed-Flow Types. Journal of Fluids Engineering, 74 (8), 1363–1380. https://doi.org/10.1115/1.4016114
22. Novak, R. A., Hearsey, R. M. (1977). A Nearly Three-Dimensional Intrablade Computing System for Turbomachinery. Journal of Fluids Engineering, 99 (1), 154–166. https://doi.org/10.1115/1.3448517
23. Davis, W. R., Millar, D. A. J. (1975). A Comparison of the Matrix and Streamline Curvature Methods of Axial Flow Turbomachinery Analysis, From a User's Point of View. Journal of Engineering for Power, 97 (4), 549–558. https://doi.org/10.1115/1.3446059
24. Denton, J. D. (1978). Throughflow Calculations for Transonic Axial Flow Turbines. Journal of Engineering for Power, 100 (2), 212–218. https://doi.org/10.1115/1.3446336
25. Li, Z., Liu, Y. (2023). An Aerodynamic Loss Model for Axial Turbine Design. Journal of Turbomachinery, 145 (9). https://doi.org/10.1115/1.4062804
26. Subbotovich, V. P., Yudin, A. Yu. (2004). Zadacha rascheta skorosti na poverhnosti lopatki turbomashiny kak zadacha optimizacii. Vestnik Nacionalnogo tehnicheskogo universiteta «HPI», 12, 101–106.
27. Subbotovich, V. P., Yudin, A. Yu., Fan, K. T. (2008). Obtekanie trehmernym potokom reshetki profilej turbomashiny na poverhnosti vrasheniya. Vestnik Nacionalnogo tehnicheskogo universiteta «HPI», 6, 41–46. Available at: https://repository.kpi.kharkov.ua/handle/KhPI-Press/18786
28. Subbotovich, V. P., Yudin, A. Yu., Temchenko, S. A. (2011). Metod rascheta techeniya v oseradialnyh kolcevyh kanalah. Visnyk Natsionalnoho tekhnichnoho universytetu «KhPI». Seriya: Enerhetychni ta teplotekhnichni protsesy y ustatkuvannia, 6, 24–27. Available at: https://library.kpi.kharkov.ua/files/Vestniki/2011_6.pdf
29. Subbotovich, V. P. (2013). Potok cherez vrashayushuyusya reshetku osevoy turbomashiny na proizvolnoy poverhnosti S1. Visnyk Natsionalnoho tekhnichnoho universytetu «KhPI». Seriya: Enerhetychni ta teplotekhnichni protsesy y ustatkuvannia, 14 (987), 43–48. Available at: https://repository.kpi.kharkov.ua/handle/KhPI-Press/3813
30. Subbotovich, V. P. (2013). Kompleksnoe teoreticheskoe i eksperimentalnoe reshenie zadach aerodinamiki protochnyh chastey turbin. Kharkiv: NTU «KhPI», 355.
31. Kopelev, S. Z., Slitenko, A. F. (1994). Konstrukcii i raschet sistem ohlazhdeniya GTD. Kharkiv: Osnova, 240.
32. Tarasov, A. I., Gurinov, A. A., Rassohin, E. V. (2005). O modelirovanii teploobmena na profilyah turbiny lopatok s pomoshyu CFD programm. Vestnik Nacionalnogo tehnicheskogo universiteta «HPI», 6, 81–84.
33. Temchenko, S. A. (2015). Obratnaya aerodinamicheskaya zadacha dlya optimalnogo proektirovaniya kolcevyh diffuzornyh kanalov turbomashin. Kharkiv: NTU «KhPI», 147.
34. Subbotovich, V. P., Yudin, A. Yu., Temchenko, S. A. (2014). Raschet turbinnoy stupeni po zazoram kak reshenie obratnyh aerodinamicheskih zadach v svobodnyh kolcevyh kanalah. Vestnik Nacionalnogo tehnicheskogo universiteta «HPI», 13, 35–38. Available at: https://repository.kpi.kharkov.ua/items/88c0b411-ad9f-4a63-b64f-2d8cf0208ae7