UDC 621.452.3.037:533.697:519.6 **doi: 10.32620/aktt.2022.4sup2.02**

Olesya DENISYUK[1], Anton BALALAIEV[2], Kateryna BALALAIEVA[2]

[1] SE Ivchenko-Progress, Zaporizhzhia, Ukraine
[2] National Aviation University, Kyiv, Ukraine

TEST PROBLEM OF THE FLOW MODELING IN AXIAL COMPRESSOR CASCADES

The flow of gas in the flow path of a gas turbine engine (GTE) is accompanied by a rather complex phenomenon. These are a three-dimensional boundary layer, an incoming vortex, a paired vortex, flow turbulence, aerodynamic wakes behind the trailing edge, separation of the boundary layer from the blade surface, pressure pulsations, uneven and unsteady flow, secondary overflows, changes in the angles of flow exit, etc. Flow R&D of a GTE remains a rather complex process, and requires the use of reliable research methods and techniques. Nowadays, two known methods are used to study a gas flow through the flow path of a GTE - experimental and calculated. Calculated, in turn, can be divided into analytical and numerical. An important stage of the numerical experiment is the solution to test problems for the possibility of setting the parameters of the numerical experiment. In this work, two test tasks were carried out. The object of the research was two compressor cascades, consisting of the identical airfoils series KR-33. The profile chord was 52 mm; the pitch cascade was 52 mm. The difference was in the installation angle of these profiles: variant 1 of the compressor cascade has an installation angle of 63.5º; variant 2 of the compressor cascade has an installation angle of 89.5º. A computational domain was constructed for each compressor cascades of airfoils and consisted of 5 million cells. Air under normal atmospheric conditions was chosen as the working fluid. The flow regime of compressor cascades varied in the range of coefficient λ = 0.26…0.9 and λ = 0.265…0.8, where the coefficient λ is the reduced velocity. The unstructured mesh method with an adaptation for the boundary layer was chosen to construct the computational mesh. Such a combination makes it possible to correctly model the flow in the boundary layer near the walls. The turbulence model SST was taken to close the Navier-Stokes equations. A comparison of the results of numerical and physical experiments for two variants of compressor cascades shows that the flow simulation error is less than 5%. Because of the calculation, the choice of this turbulence model for subsequent studies of the flow in the stages of the compressor, fan, and propfan will be justified.

Keywords: *test problem; compressor cascade; numerical simulation; axial compressor; installation angle; unstructured mesh.*

Introduction

A gas flow in the flow path of a gas turbine engine (GTE) is a rather complicated process. It is well described by the system of equations of continuum mechanics. These equations are too complex to have an analytical solution. However, there is a large class of gas dynamics problems in which it is permissible to use various simplified mathematical models of gas flow. Therefore, when reducing the number of three-dimensional coordinates to a one-dimensional model and some other simplifications, we get a system of equations that can be easily solved analytically. For example, when studying a gas flow in the flow path of an axial compressor stage, the flow is cut by the current surfaces into a number of axisymmetric layers with infinitely small thickness. Such a stage element is called an elementary stage. It is convenient to deploy an elementary stage onto a plane. The flow in an elementary stage is considered as a flat flow. Thus, we have obtained a two-dimensional model of a gas flow. All parameters of a gas flow (speed, pressure, temperature, density) can be averaged in the direction parallel to the front of the blades, that is, at the compressor inlet, at the compressor outlet and in the gap. This means that, for example, the speed at the entrance to the impeller at any place along the front is constant in magnitude and direction. Such a flow at the elementary level is considered as one-dimensional, and its analysis is so simplified that it involves the use of analytical research methods.

Despite its simplicity, this model allows analytical tools to obtain quickly a number of important calculation data required for engine design. Therefore, analytical methods are widely used in the practice of aircraft engine building. However, they require the involvement of too rough simplifications of a physical nature and it limits the field of application of the method. Such methods can only be used for calculating the flow in channels of simple shapes and under idealized flow conditions.

The above properties of analytical methods indicate that such methods are needed at the initial stages of

ISSN 1727-7337 (print)
ISSN 2663-2217 (online)

designing GTE when it is necessary to determine the main parameters of the design engine. As a result of using such methods, the design of the GTE will have a number of disadvantages.

1. Analysis of previous research and publications

In the past few decades, interest has increased in the use of computational methods to solve the gas dynamics of turbomachines. Due to a significant increase in computing resources, it allows you to solve complex systems of equations describing physical phenomena quickly and efficiently. The cost of an experimental study of complex technical systems can exceed the cost of their calculations by orders of magnitude. In some cases, it is not possible to see the flow pattern using experimental studies. In this case, the use of computational methods is practically the only research tool.

The solution to any problem of numerical simulation can be represented in the form of the following components [1]:

1. Construction of a mathematical model and its justification. Typically, a mathematical model includes complex partial differential equations. This stage includes several sub-items:

- formulation of the problem;
- selection of a numerical scheme;
- creation a mesh and choosing a time step;
- construction of a numerical scheme.

The mesh cells are divided into groups. Boundary conditions are set for each group of cells.

2. Solving the equations of the mathematical model.

3. Analysis of the results obtained, comparing them with the experimental results or other calculations.

Scientific and technical problems, that are being solved, can be conditionally divided into two groups, according to the adequacy of mathematical models, which describe:

- problems for which mathematical models are sufficiently substantiated and have been verified;
- problems for which the reliability of mathematical models has yet to be proven.

The system of Navier-Stokes differential equations describes the gas flow. The system of equations is rather complicated and it is still impossible to solve it analytically. Therefore, in the practice of studying gas flows numerical methods for solving such equations are widely used. For this, the solid computational domain through which the gas flows is represented as a set of isolated nodes or as a mesh. It means that the solid area appears as a discrete one. The equations are solved only at those nodes where the system finite-difference analogue of Navier-Stokes is used (all partial derivatives are replaced by finite differences). Therefore, each differential equation is written as a linear algebraic equation. Thus, solving a system of differential equations for a continuous computational domain through which the gas flows is reduced to solving a system of algebraic equations at each mesh node.

The solution to the system of these equations is not a solution to the system of differential equations because of some simplifications, which are used in constructing of its finite-difference analogue.

The task of constructing a computational mesh is to find a mapping that translates the mesh nodes in the physical area into the computational one. At least this reflection should correspond the following requirements [1-6]:

1. The reflection should be unambiguous.
2. The mesh should have a refinement in the computational areas, where large gradients of functions may appear.
3. The mesh lines should be smooth to ensure continuity of derivatives.
4. The mesh should be as close to orthogonal as possible (the boundaries of the mesh elements should intersect at the angles close to 90º).
5. The aspect ratio of the mesh element should not be too large (ideally, it should equal one).

If the set of network nodes of the computational mesh is ordered (i.e. it can be specified in the form of some function of coordinates), then such a mesh is called regular or structured.

If the location of network nodes of the computational mesh does not obey any specific law, then such a mesh is called unstructured.

The use of structured meshes, in comparison with unstructured meshes, makes it possible to reduce the computation time and the required amount of computer memory. At the same time, the procedure for constructing a curvilinear regular mesh in the general case is rather complicated operation that requires a lot of time compared to the procedure for constructing an irregular mesh.

Choosing a meshing method (structured or unstructured), consider the following factors:

1. Structured meshes allow a higher order of approximation than unstructured ones.
2. A streamline with strong shock waves is better solved on structured meshes than on unstructured ones.
3. Programs with structured meshes are simpler, since they do not require storing and processing information about neighboring cells (orientation, length, etc.), which are necessary when calculating on unstructured meshes.
4. The task of constructing structured meshes for bodies of complex geometry is very laborious, in addi-

tion, degenerate cells may appear, which leads to a significant decrease in accuracy.

5. A significant advantage of the unstructured approach is a flexible mesh generation structure that allows you to display the geometry of the computational domain accurately and generate a mesh at a lower cost for areas of complex geometry, mainly three-dimensional configurations.

6. The adaptation of the mesh to the solution of the problem in the case of the unstructured approach is relatively simpler than in the case of structured methods of mesh construction.

To obtain a solution to any partial differential equation, it is necessary to set conditions on the boundary of the considered region [1, 7-9].

Boundary conditions can be classified mathematically and physically.

From a mathematical point of view, there are three types of boundary conditions:

1. Dirichlet condition defines value of the quantity at the boundary by which the equation is written.

2. Neumann condition, according to which it is implied that the derivative of the function is set at the boundary of the desired computational domain.

3. Mixed boundary conditions are defined as a linear combination of Dirichlet and Neumann conditions.

From a physical point of view, boundary conditions are divided into the following types:

1. Limiting conditions at the inlet.

2. Limiting conditions on the wall.

3. Limiting conditions of symmetry. Such conditions apply to problems in which symmetry exists. The computational area is limited by the line (plane) of symmetry, which allows reducing the computation time and computer resources.

4. Periodic boundary conditions. These conditions are similar to limiting conditions of symmetry. They are used to calculate currents where there are many identical sections. Instead of looking at the entire flow pattern, only one section is considered (for example, the flow in blade machines).

At present, the main approach to calculating turbulent flows is the numerical solution of the averaged Navier-Stokes equations. These equations are also called the Reynolds equations [10].

Time-averaging leads to appearing new components in the equations, which can be interpreted as gradients of “apparent” (additional) stresses and heat flows associated with turbulent motion. These new quantities should be related to the characteristics of the averaged flow, using turbulence models, which leads to the emergence of new hypotheses and approximations.

According to one of the classifications, turbulence models are divided into several classes [8, 10-11]:

1. Zero-order models are models that contain partial differential equations for the mean velocity field only.

2. One-parameter models are models that use partial differential equations for the scale of turbulent flow velocity in addition to partial differential equations for averaging motion.

3. Two-parameter models are models that include, in addition to partial differential equations for averaged motion and partial differential equations for the scale of turbulent flow velocity, a partial differential equation for the linear length of the turbulent flow.

4. Reynolds stress models are models that consist of partial differential equations for all components of the Reynolds stress and for the length scale in general.

Purpose of the work – developing test problem of the flow modeling in axial compressor cascade.

2. Materials and Methods

The method that used in this work is numerical experiment. The results of numerical experiment are compared with the results of physical experiment. The unstructured mesh method with adaptation for the boundary layer was chosen for constructing the computational mesh. Such combination makes it possible to model the flow in the boundary layer near the walls correctly. The turbulence model SST was taken to close the Navier-Stokes equations. As a result of the calculation the choice of this turbulence model for subsequent studies of the flow in the stages of the compressor, fan, and propfan will be justified.

3. Results

The object of the research was two compressor cascades, consisting of the identical airfoils series KR-33. The profile chord was 52 mm; the pitch cascade was 52 mm. The difference was in the installation angle of these profiles.

A computational domain was constructed for each compressor cascades of airfoils and consisted of 5 million cells. Air under normal atmospheric conditions was chosen as the working fluid. The flow regime of compressor cascades varied in the range of coefficient λ = 0.26…0.9 and λ = 0.265…0.8, where the coefficient λ is the reduced velocity that was defined as the ratio of the axial air flow velocity at the entrance to the cascade to the sound velocity.

The degree of pressure increase of the compressor stage π, determined by the total pressure, was calculated as the result of the numerical experiment.

Fig. 1, 2 show the visualization (the velocity field) of the flow around the considered compressor cascades (variant 1 and variant 2, respectively). Variant 1 of the

ISSN 1727-7337 (print)
ISSN 2663-2217 (online)

compressor cascade has an installation angle of 63.5º; variant 2 of the compressor cascade has an installation angle of 89.5º.

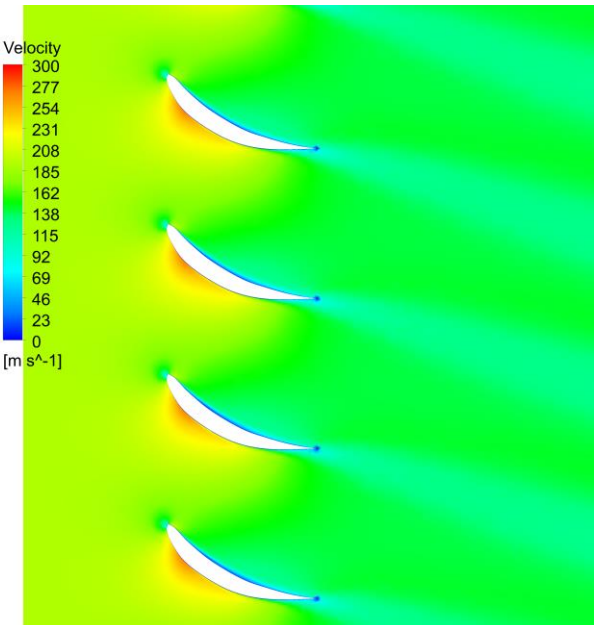

Fig. 1. The velocity field in the flow around the compressor cascade (variant 1) at λ =0.6

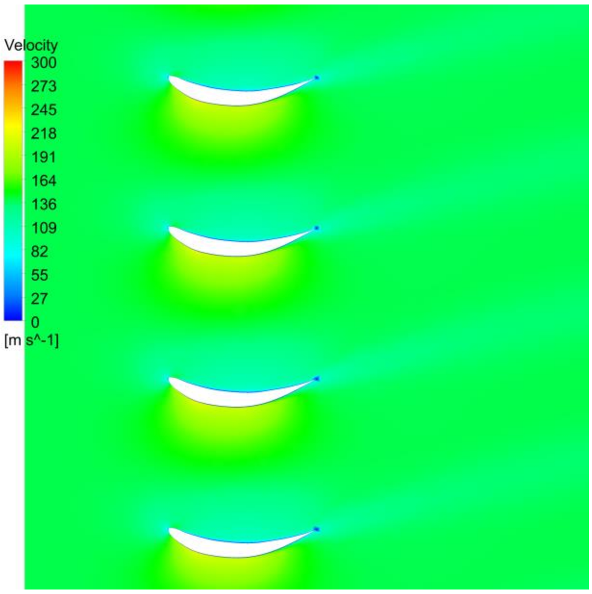

Fig. 2. The velocity field in the flow around the compressor cascade (variant 2) at λ =0.445

Fig. 3, 4 show dependence of the total pressure loss coefficient δ on the reduced velocity λ (variant 1 and variant 2, respectively): on the graphs the lines represent the results of numerical experiment and the dots reflect the results of physical experiment [12].

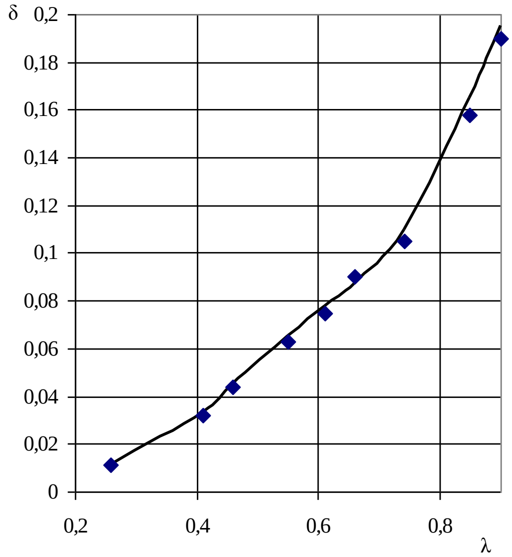

Fig. 3. Dependence of the total pressure loss coefficient δ on the reduced velocity λ (variant 1)

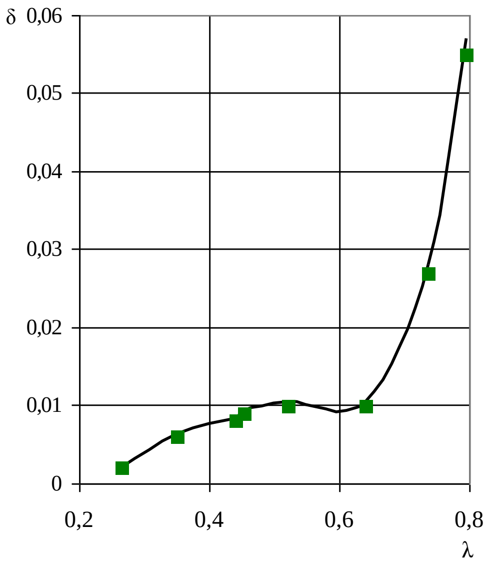

Fig. 4. Dependence of the total pressure loss coefficient δ on the reduced velocity λ (variant 2)

Comparison of the results of numerical and physical experiments for two variants of compressor cascades shows that the flow simulation error is less than 5%.

The calculation error is 2.3...4.6 % for variant 1 of the compressor cascade with an installation angle of 63.5º for the considered range of coefficient λ=0.26...0.9.

The calculation error is 3.7…4.9% for variant 2 of the compressor cascade with an installation angle of 89.5º for the considered range of coefficient λ=0.265…0.8.

Conclusions

The article presents the results of a test problem of flow modeling in two versions of compressor cascades with coefficient λ=0.26…0.9 and coefficient λ=0.265…0.8. Comparison of the results of numerical calculations with experimental data showed that when using SST turbulence models to close the Navier-Stokes equations, the calculation error was up to 5%. Thus, it is planned to use the SST turbulence model in further calculations of the flow in the stages of axial compressor, fan, and propfan.

References (GOST 7.1:2006)

1. Current Trends and Open Problems in Computational Mechanics [Text] / F. Aldakheel, B. Hudobivnik, M. Soleimani [et al.]. – Cham. : Springer. – 2022. – 607 p. DOI: 10.1007/978-3-030-87312-7.

2. Аэродинамический расчет и оптимальное проектирование проточной части турбомашин [Текст] / А. В. Бойко, Ю. Н. Говорущенко, С. В. Ершов [и др.]. – Х. : НТУ «ХПИ», 2002. – 356 с.

3. Бойко, А. В. Применение вычислительной аэродинамики к оптимизации лопаток турбомашин [Текст] / А.В. Бойко, Ю. Н. Говорущенко, М. В. Бурлака. – Х. : НТУ «ХПИ», 2013. – 193 с.

4. Ferziger, J. H. Computational Methods for Fluid Dynamics [Text] / Joel H. Ferziger, Milovan Perić, Robert L. Street. – Cham. : Springer, 2020. – 596 p. DOI: 10.1007/978-3-319-99693-6.

5. Ершов, С. В. О выборе степени измельчения сетки при расчетах трехмерных течений вязкого газа в турбомашинах [Текст] / С. В. Ершов, В. А. Яковлев // Вісник двигунобудування. – 2015. – № 2. – С. 171–176.

6. Moukalled, F. The Finite Volume Method in Computational Fluid Dynamics [Text] / F. Moukalled, L. Mangani, M. Darwish. – Cham. : Springer, 2016. – 791 p. DOI: 10.1007/978-3-319-16874-6.

7. Numerical Simulation of Turbulent Flows and Noise Generation [Text] / Ch. Brun, D. Juvé, M. Manhart, C.-D. Munz. – Berlin : Springer-Verlag, 2009. – 342 p. DOI: 10.1007/978-3-540-89956-3.

8. Zeidan, D. Advances in Fluid Mechanics [Text] / D. Zeidan, L. T. Zhang, E. G. Da Silva, J. Merker. – Singapore : Springer, 2022. – 232 p. DOI: 10.1007/978-981-19-1438-6.

9. Berselli, L. C. Mathematics of Large Eddy Simulation of Turbulent Flows [Text] / Luigi C. Berselli, Traian Iliescu, William J. Layton. – Berlin : Springer, 2006. – 350 p. DOI: 10.1007/b137408.

10. Rodriguez, S. Applied Computational Fluid Dynamics and Turbulence Modeling [Text] / Sal Rodriguez. – Cham. : Springer, 2019. – 306 p. DOI: 10.1007/978-3-030-28691-0.

11. Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environments by CFD: Part 1 – Summary of Prevalent Turbulence Models [Text] / Zhiqiang John Zhai, Zhao Zhang, Wei Zhang, Qingyan Yan Chen // J. HVAC&R Research. – 2007. – Vol. 13, No. 6. – P. 853–870.

12. Свечников, В. С. Аэродинамические характеристики решетки осевых компрессоров [Текст] / В. С. Свечников, А. Б. Кириллов. – М. : ЦАГИ, 1958. – 94 с.

References (BSI)

1. Aldakheel, F., Hudobivnik, B., Soleimani, M., Wessels, H., Weißenfels, Ch., Marino, M. Current Trends and Open Problems in Computational Mechanics. *Springer*, 2022, 607 p. DOI: 10.1007/978-3-030-87312-7.

2. Bojko, A. V., Govorushhenko, Ju. N., Ershov, S. V., Rusanov, A. V., Severin, S. D. A*jerodinamicheskij raschet i optimal'noe proektirovanie protochnoj chasti turbomashin* [Aerodynamic calculation and optimal design of the flow path of turbomachines]. Kharkov, NTU "KhPI", 2002. 356 p.

3. Bojko, A. V., Govorushhenko, Ju. N., Burlaka, M. V. *Primenenie vychislitel'noj ajerodinamiki k optimizacii lopatok turbomashin* [Application of computational aerodynamics to optimization of turbomachine blades]. Kharkov, NTU "KhPI", 2013. 193 p.

4. Ferziger, J. H., Perić, M., Street, R. L. *Computational Methods for Fluid Dynamics*. Springer, 2020, 596 p. DOI: 10.1007/978-3-319-99693-6.

5. Ershov, S. V., Jakovlev, V. A. O vybore stepeni izmel'chenija setki pri raschetah trehmernyh techenij vjazkogo gaza v turbomashinah [On the Choice of the Degree of Grid Refinement in the Calculation of Three-Dimensional Viscous Gas Flows in Turbomachines]. *Visnik dvigunobuduvannja - Bulletin of Engine Building*, 2015, no. 2, pp. 171–176.

6. Moukalled, F., Mangani, L., Darwish, M. *The Finite Volume Method in Computational Fluid Dynamics*. Springer, 2016. 791 p. DOI: 10.1007/978-3-319-16874-6.

7. Brun, Ch., Juvé, D., Manhart, M., Munz, C.-D. *Numerical Simulation of Turbulent Flows and Noise Generation*. Springer-Verlag, 2009. 342 p. DOI: 10.1007/978-3-540-89956-3.

8. Zeidan, D., Zhang, L.T., Da Silva, E.G., Merker, J. *Advances in Fluid Mechanics*. Springer, 2022. 232 p. DOI: 10.1007/978-981-19-1438-6.

9. Berselli, L. C., Iliescu, T., Layton, W. J. *Mathematics of Large Eddy Simulation of Turbulent Flows*. Springer, 2006. 350 p. DOI: 10.1007/b137408.

10. Rodriguez, S. *Applied Computational Fluid Dynamics and Turbulence Modeling*. Springer, 2019. 306 p. DOI: 10.1007/978-3-030-28691-0.

11. Zhai, Z. J., Zhang, Z., Zhang, W., Chen, Q. Y. Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environments by

ISSN 1727-7337 (print)
ISSN 2663-2217 (online)

CFD: Part 1 – Summary of Prevalent Turbulence Models. *Journal HVAC&R Research,* 2007, vol. 13, no. 6, pp. 853–870.

12. Svechnikov, V. S., Kirillov, A. B. *Ajerodinamicheskie harakteristiki reshetki osevyh kompressorov* [Aerodynamic characteristics of the axial compressors' cascade]. Moscow, TsAGI Publ., 1958. 94 p.

Надійшла до редакції 12.06.2022, розглянута на редколегії 8.08.2022

ТЕСТОВА ЗАДАЧА МОДЕЛЮВАННЯ ТЕЧІЇ В КОМПРЕСОРНИХ РЕШІТКАХ

О. В. Денисюк, А. В. Балалаєв, К. В. Балалаєва

Течія газу в проточній частині газотурбінного двигуна (ГТД) супроводжується досить складними явищами. Це тривимірний прикордонний шар, вхідний вихор, парний вихор, турбулентність потоку, аеродинамічні сліди за задньою кромкою лопатки, відрив прикордонного шару від поверхні лопатки, пульсації тиску, нерівномірність і нестаціонарність течії, вторинні перетікання, зміна кутів виходу потоку і т. д. Дослідження течії та розробка ГТД залишаються досить складними процесами, що вимагають застосування надійних методів та методик дослідження. На сьогоднішній день відомі два методи дослідження течії газу в проточній частині ГТД - експериментальний та розрахунковий. Розрахункові, в свою чергу, можна поділити на аналітичні та числові. Важливим етапом чисельного експерименту є вирішення тестових завдань для можливості налаштування параметрів чисельного експерименту. У цій роботі було проведено дві тестові задачі. Об'єктом дослідження були два компресорні решітки, що складаються з однакових профілів серії КР-33 з хордою профілю 52 мм і кроком решітки 52 мм. Відмінність полягала у куті установки цих профілів: перший варіант компресорної решітки мав кут установки 63,5º, а другий варіант – 89,5º. Розрахункова область будувалася для кожної компресорної решітки профілів і складалася з 5 млн комірок. Як робоче тіло було обрано повітря за нормальних атмосферних умов. Режим течії в компресорних решітках варіювався в межах коефіцієнтів λ=0,26…0,9 та λ=0,265…0,8, де коефіцієнт λ – приведена швидкість. Для побудови розрахункової сітки було обрано метод неструктурованих сіток із адаптацією до прикордонного шару. Таке поєднання дозволяє коректно моделювати течію у прикордонному шарі біля стінок. Для замикання рівнянь Нав'є-Стокса було взято модель турбулентності SST. Порівняння результатів чисельних і фізичних експериментів для двох варіантів компресорних решіток показує, що похибка моделювання течії становить менше 5%. В результаті розрахунку обґрунтовано вибір даної моделі турбулентності для подальших досліджень течії в ступенях компресора, вентилятора та гвинтовентилятора.

Ключові слова: тестова задача; решітка профілів; чисельне моделювання; осьовий компресор; кут установки лопатки; неструктурована сітка.

Денисюк Олеся Валеріївна – інженер-конструктор 1 категорії, ДП «Івченко-Прогрес», Запоріжжя, Україна.

Балалаєв Антон Валерійович – канд. техн. наук, старш. викл. каф. прикладної механіки та інженерії матеріалів, Національний авіаційний університет, Київ, Україна.

Балалаєва Катерина Вікторівна – д-р техн. наук, доц., проф. каф. авіаційних двигунів, Національний авіаційний університет, Київ, Україна.

Olesya Denisyuk – engineer-designer of 1 category, SE Ivchenko-Progress, Zaporizhzhia, Ukraine,
e-mail: Denisyukolesya@gmail.com, ORCID: 0000-0001-7516-7399.

Anton Balalaiev – Assistant of Mechanics Department, National Aviation University, Kyiv, Ukraine,
e-mail: avbalalaev@ukr.net, ORCID: 0000-0003-3603-4512, Scopus Author ID: 56955689800.

Kateryna Balalaieva – Doctor of Technical Sciences, Associate Professor, Professor of Dept. of aviation engine, National Aviation University, Kyiv, Ukraine,
e-mail: Kiki_ua@ukr.net, ORCID: 0000-0001-6495-3263, Scopus Author ID: 57190439468.