UDC 629.735.03:621.43.031.3(045)
DOI: 10.15587/1729-4061.2017.101315

# NUMERICAL STUDY OF FLOW IN THE STAGE OF AN AXIAL COMPRESSOR WITH DIFFERENT TOPOLOGY OF COMPUTATIONAL GRID

**Yu. Tereshchenko**
Doctor of Technical Sciences, Professor*
E-mail: Terj@nau.edu.ua
**E. Doroshenko**
PhD*
E-mail: Kiki_ua@ukr.net
**I. Lastivka**
Doctor of Technical Sciences,
Professor, Head of Department
Department of Higher Mathematics**
E-mail: Iola@nau.edu.ua
**Yu. Tereshchenko**
PhD*
E-mail: Terj-nau@ukr.net
*Department of Aviation Engine**
**National Aviation University
Kosmonavta Komarova ave., 1, Kyiv, Ukraine, 03058

*Представлено результати чисельного моделювання течії в ступені осьового компресора. Порівняння результатів чисельного експерименту з даними фізичних досліджень показало, що похибка розрахункового дослідження становить 0.3...8.6 %. На основі аналізу результатів чисельного експерименту встановлено, що для вирішення задач внутрішньої аеродинаміки компресорів доцільним є використання моделі турбулентної в'язкості SST і дрібної адаптивної сітки*

*Ключові слова: моделювання течії, розрахункова сітка, модель турбулентної в'язкості, ступінь компресора, примежевий шар*

*Представлены результаты численного моделирования течения в ступени осевого компрессора. Сравнение результатов численного эксперимента с данными физических исследований показало, что погрешность расчетного исследования составляет 0.3...8.6 %. На основе анализа результатов численного эксперимента установлено, что для решения задач внутренней аэродинамики компрессоров предпочтительным является использование модели турбулентной вязкости SST и мелкой адаптивной сетки*

*Ключевые слова: моделирование течения, расчетная сетка, модель турбулентной вязкости, ступень компрессора, пограничный слой*

## 1. Introduction

Axial compressors are one of the main elements of modern gas-turbine engines. Improving the efficiency and cost-effectiveness of gas turbine engines largely depends on solving the problem of improvement of internal aerodynamics of axial compressors [1]. Improvement of the parameters and characteristics of compressors is based on the results of studies that are conducted in two directions.

The first direction provides obtaining of reliable methods of aerodynamic design, based on the results of physical modeling of processes in the flow part of axial compressors.

The second direction implies receiving reliable methods of aerodynamic design of compressors based on the use of methods of mathematical modeling to ensure obtaining the assigned parameters of compressors without lengthy and labor-consuming adjusting operations.

Solving the problems according to the second direction implies in turn the solution of direct and inverse problems of gas dynamics of turbo-machines. Solving the direct problem provides obtaining reliable methods of aerodynamic design of compressors in order to receive the assigned parameters. Solution of the inverse problem implies development of computational methods, obtaining and analyzing the characteristics of compressors by the known geometry of the flow part.

Experimental methods of research do not always allow for the visualization and analysis of the flow picture in a compressor [1, 2]. That is why at present in order to devise and examine the ways of aerodynamic improvement of axial compressors, numerical techniques for modeling a flow in compressors are applied.

When employing a numerical experiment, one of the key issues is a substantiated choice of the model of turbulent viscosity and the topology of computational grid.

## 2. Literature review and problem statement

Addressing the aerodynamics of compressors has been the subject of a large number of theoretical and experimental studies. Of particular interest is the research into flow in the stages of axial compressors by using the numerical experiment that makes it possible to analyze the qualitative picture of flow-over in the inter-blade channels under all operating modes. Article [3] gives a literary review of studies into the use of methods of computational fluid dynamics (CFD) to analyze the flow in turbo machines. The problems are considered that occur when employing numerical modeling, and the advantages and shortcomings of CFD analysis in turbo machines are analyzed. Materials of the articles reveal general tendencies of using a numerical experiment in turbo machines. Numerical experiment allows exploring different modes of the flow in compressors. Paper [4] presents results of modeling the flow and the evaluation of influence of non-uniform flow at the inlet to the compressor on the stability of the flow. However, the paper lacks outlining the method of calculating a limit of the stable operation of a compressor. Articles [5, 6] describe results of the numerical simulation of flow under critical modes of operation of compressor grids. The authors considered the question of determining the optimal parameters of multiserial compressor grids by using mathematical modeling. However, these studies did not take into account the three-dimensional effects of flow-over when contructing mathematical models. Authors of article [7], by using a numerical experiment, considered the problem on the optimization of compressor blade shape in order to improve efficiency of the compressor. Paper [8] examined the problem of calculating the characteristics of an axial compressor. However, it lacks a sufficient substantiation for the choice of a computational grid.

When solving problems on the numerical simulation of flow in the elements of a compressor, various models of turbulent viscosity are applied. In article [9], the model of turbulent viscosity SST closes the Navier-Stokes equations, but there is no comparison with the results obtained when employing the model of turbulent viscosity k–ε. Paper [10] proposed using a modified model of turbulent viscosity k–ω, which provides for a high accuracy of computational study into the flow in the elements of axial compressor. However, the use of the modified model of turbulence k–ω is complicated by a change in the programming code. In article [11], authors presented results on choosing a model of turbulent viscosity when calculating the pressure losses in the flow part of a gas turbine engine. Between the considered models (SST and k–ε), choosing a model of turbulent viscosity k–ε was substantiated. The calculations, however, were carried out for a computational grid without adaptation of the near-border layer.

Thus, we can state that numerical experiment is widely used to study the flow in the elements of a compressor. Nevertheless, results of articles [4–11] reveal that in order to solve various problems on modeling the flow in the elements of compressors, it is expedient to apply various models of turbulent viscosity and the topology of a computational grid. That is why solving a particular class of problems requires preliminary setting of the numerical experiment, which means performing the test tasks.

## 3. The aim and tasks of research

The aim of present work is the comparative assessment of accuracy of different topologies of computational grid and the models of turbulent viscosity in order to model the flow in the stage of an axial compressor.

To accomplish the aim, the following tasks are to be solved:

– to create a 3D model of the stage of an axial compressor at different variants of topology of the computational grid;

– to perform a computational study and evaluate the accuracy of calculations;

– to substantiate the application of feasible variants of topologies of computational grid and the models of turbulent viscosity when calculating the flow in the stage of an axial compressor.

## 4. Method of examining the flow in the stage of an axial compressor

To study the flow in the stage of an axial compressor, we have chosen the method of numerical experiment. A numerical experiment includes the following stages [12]:

– construction of a geometric model;
– construction of the computational grid;
– determining the initial and boundary conditions;
– numerical solution of the problem;
– visualization of the solution of the problem;
– analysis of the solution.

In order to conduct the research, we employed in the stage of an axial compressor the coarse, fine and fine adaptive unstructured computational grid.

Calculation of turbulent flow of gas was carried out by numerical solution of the averaged Navier-Stokes equations. During numerical simulation of the flow, we applied the turbulent viscosity model SST [13] and k-ε [14].

Accuracy of the results obtained was achieved due to using the verified methods of calculation and was assessed by aligning the results of calculations with data of experimental studies by other authors [15].

## 5. Results of examining the modeling of the flow in the stage of a compressor

We chose as the object of study a stage of the axial compressor (Fig. 1) that consists of an inlet guide unit (IGU), an impeller (Im) and a guiding device (GD). Blade crown of BHA consists of 30 blades; Im and GD have 24 blades each.

Fig. 2 shows schematic and basic geometric parameters of subsonic grid of the blade crown of an axial compressor.

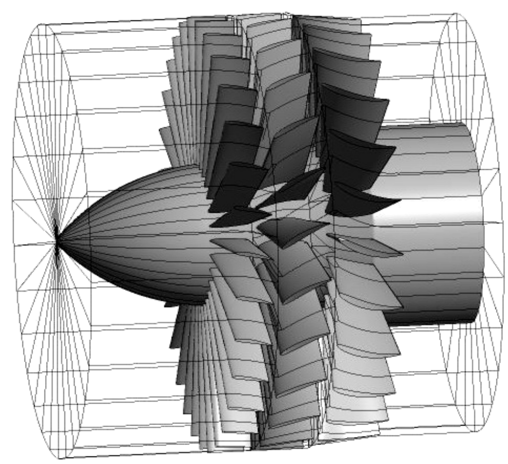


Fig. 1. 3D model of the stage of an axial compressor

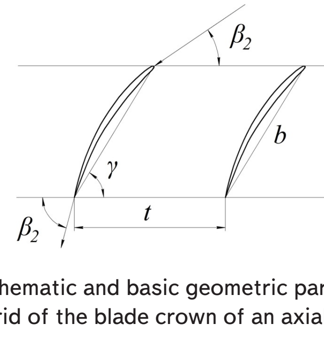



Fig. 2. Schematic and basic geometric parameters of subsonic grid of the blade crown of an axial compressor

Table 1 below gives geometric characteristics of the blade crowns: r is the radius of the cross-section, mm; t/b is the relative pitch of the lattice; $\beta_1$ is the angle of flow inlet; $\beta_2$ is the angle of flow outlet; $\gamma$ is the angle between the chord of profile b and the plane of rotor rotation.

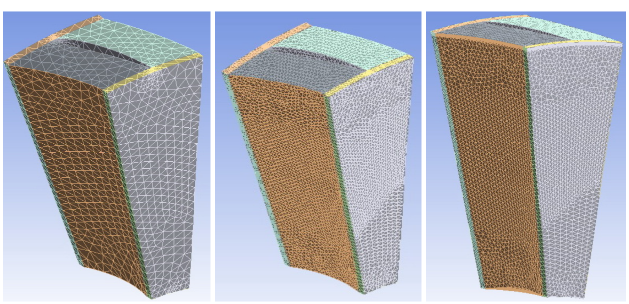
Table 1

Geometric characteristics of the of the blade crowns

| Inlet guide unit | | | | | |
|---|---|---|---|---|---|
| r, mm | 80 | 104 | 136 | 168 | 200 |
| t/b | 0.48 | 0.624 | 0.816 | 1.008 | 1.2 |
| $\gamma$ | 89° 18' | 87° 37' | 85° 10' | 83° | 81° 46' |
| Impeller | | | | | |
| r, mm | 80 | 104 | 136 | 168 | 200 |
| t/b | 0.468 | 0.589 | 0.742 | 0.878 | 1.005 |
| $\beta_1$ | 62° 37' | 57° 55' | 51° 47' | 46° 42' | 42° 09' |
| $\beta_2$ | 101° 43' | 96° 27' | 88° 30' | 80° 03' | 71° 13' |
| $\gamma$ | 85° 01' | 80° 23' | 73° 40' | 66° 57' | 60° 02' |
| Guiding device | | | | | |
| r, mm | 80 | 104 | 136 | 168 | 200 |
| t/b | 0.475 | 0.55 | 0.633 | 0.699 | 0.747 |
| $\gamma$ | 76° 30' | 74° 40' | 72° 17' | 69° 59' | 67° 36' |

To employ the frequency condition properly, the estimated area consisted of one blade and an IGU inter-blade channel, one blade and an Im inter-blade channel, one blade and a GD inter-blade channel.

In the present study, we carried out a series of calculations with the models of turbulent viscosity k–ε, SST and three variants of three-dimensional unstructured computational grids. We examined the coarse computational grid, fine computational grid and fine adaptive computational grid (Fig. 3).



*a* *b* *c*

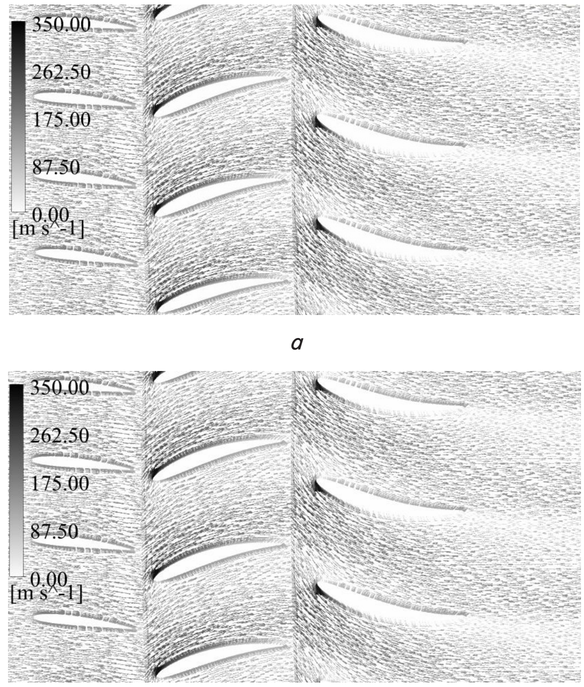
Fig. 3. Variants of computational grid for the impeller of the stage of an axial compressor: *a* – coarse computational grid; *b* – fine computational grid; *c* – fine adaptive computational grid

Computational area for the coarse grid comprised 162 thousand cells, for the fine grid – 1.7 million cells, for fine adaptive – 2.013 million cells.

To simulate the flow, we chose a second order computational scheme with a local use of the first order computational scheme (High resolution). This is the most accurate calculation scheme for the given class of problems, which is featured in the software module [16].

For each variant of the computational grid, we calculated each of the two models of turbulent viscosity at axial velocity at input from 110 to 150 m/s.

Fig. 4 shows instantaneous vector velocity field on the mean radius of the stage of an axial compressor for the coarse grid (variant No. 1) for two models of turbulent viscosity at axial velocity at input of 130 m/s.




*b*

Fig. 4. Instantaneous vector velocity field on the mean radius of the stage of an axial compressor for the coarse grid: *a* – model of turbulent viscosity k–ε; *b* – model of turbulent viscosity SST

In Fig. 4, qualitative picture of flow-over at various models of turbulent viscosity does not differ.

Fig. 5, 6 show the instantaneous vector velocity field on the mean radius of the stage of an axial compressor for the fine grid and the fine adaptive grid for two models of turbulent viscosity. Axial velocity at input made up 130 m/s.

Fig. 5, 6 demonstrate that the picture of flow-over has a different character when modeling a flow with different models of turbulence for the fine and the fine adaptive calculation grids.

In order to assess the accuracy of computational research, in the course of present work we performed a comparative analysis of results of the physical [15] and numerical experiments.

Fig. 7, 8 show a dependence of the degree of pressure increase in the stage of an axial compressor on the velocity coefficient.

A degree of pressure increase is determined by formula:

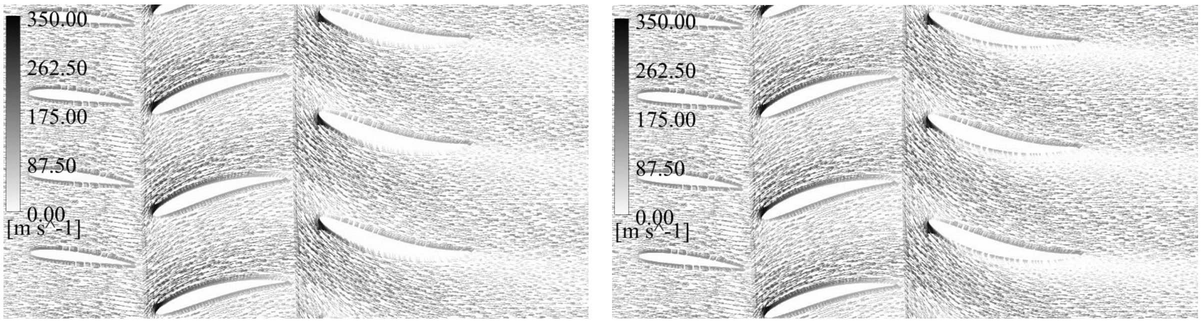
$$\pi = \frac{p_2}{p_1}, \tag{1}$$

where $p_1$ is the full pressure at the input to the stage; $p_2$ is the full pressure at the output of the stage.

Velocity coefficient is determined by ratio:

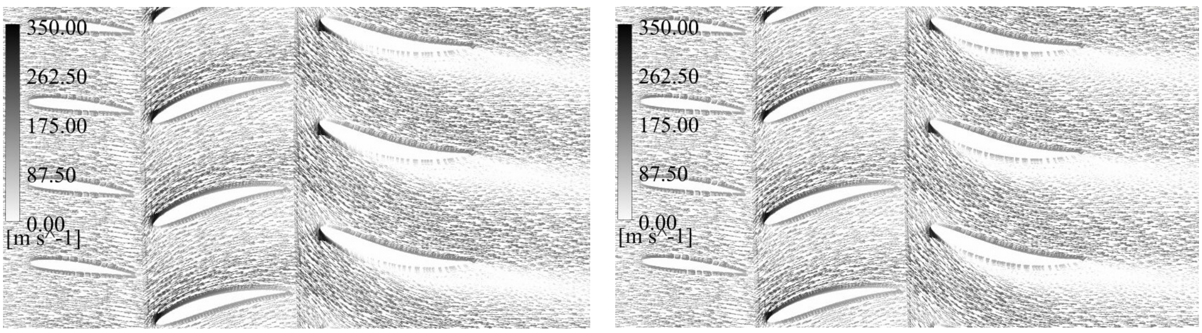
$$\lambda = \frac{c}{a}, \tag{2}$$

where c is the axial flow rate at the input, a is the speed of sound.




*a* *b*

Fig. 5. Instantaneous vector velocity field on the mean radius of the stage of an axial compressor for the fine grid: *a* – model of turbulent viscosity k–ε; *b* – model of turbulent viscosity SST




*a* *b*

Fig. 6. Instantaneous vector velocity field on the mean radius of the stage of an axial compressor for the fine adaptive grid: *a* – model of turbulent viscosity k–ε; *b* – model of turbulent viscosity SST

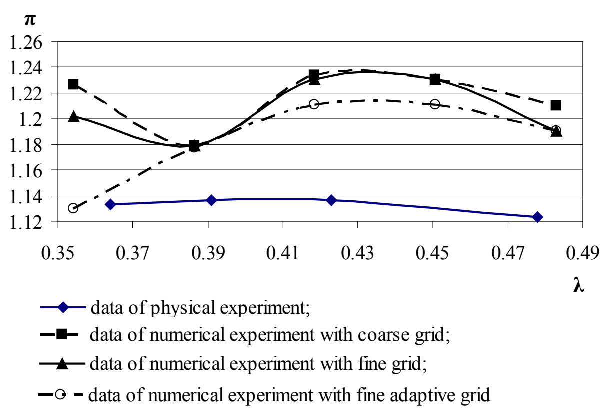



Fig. 7. Dependence of the degree of pressure increase in the stage of an axial compressor $\pi$ on the velocity coefficient $\lambda$ when calculating using the model of turbulent viscosity SST

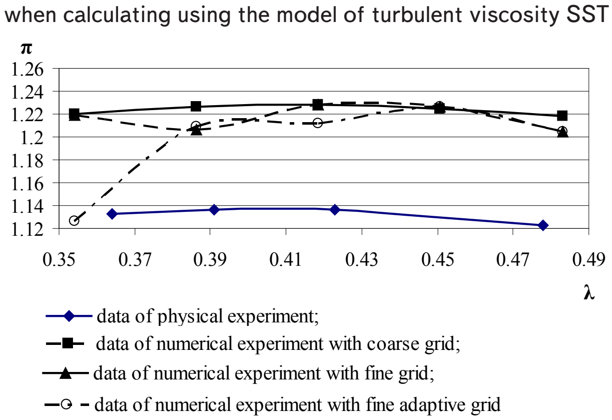



Fig. 8. Dependence of the degree of pressure increase in the stage of an axial compressor $\pi$ on the velocity coefficient $\lambda$ when calculating using the model of turbulent viscosity k–ε

For a more detailed comparison, Fig. 9 shows results of numerical calculation for two models of turbulent viscosity for the fine adaptive computational grid.

Graphical dependences in Fig. 9 show that the model of turbulent viscosity SST yields a smaller error when calculating the parameters of flow in the stage of an axial compressor.

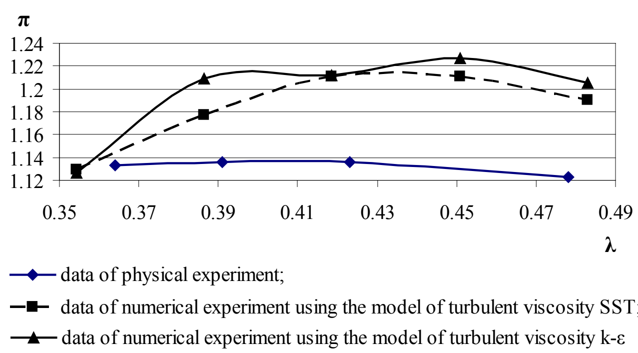



Fig. 9. Dependence of the degree of pressure increase in the stage of an axial compressor $\pi$ on the velocity coefficient $\lambda$ for two models of turbulent viscosity

## 6. Discussion of results of examining the modeling of flow in the stage of an axial compressor

Simulation of the flow was carried out for the stage of an axial compressor with parameters, similar to those of the stage D-1 [15]. Results of the conducted numerical modeling of the flow for two models of turbulent viscosity and three variants of computational grids allow us to compare the accuracy at computation. The accuracy of results of the computational research into the simulation of flow in the stage of an axial compressor was assessed by an error using the dependence:

$$\delta = \left| \frac{\pi_e - \pi}{\pi_e} \right| \cdot 100\ \%, \tag{3}$$

where $\pi_e$ is the degree of pressure increase by the results of physical experiment [15], $\pi$ is the degree of pressure increase received by the results of computational research.

Flow simulation using the model of turbulent viscosity k–ε has the following error:

– for the coarse grid 7.7–8.5 %;
– for the fine grid 7.6–8.5 %;
– for the fine adaptive grid 0.6–8.4 %.

An error of calculation for the simulation of flow in the stage of a compressor using the model of turbulent viscosity SST is:

– for the coarse grid 3.8–8.6 %;
– for the fine grid 3.8–8.5 %;
– for the fine adaptive grid 0.3–6.5 %.

According to the calculations, the models of turbulent viscosity SST and k–ε display the errors of one order.

However, in order to solve the problems of internal aerodynamics of the compressors, it is necessary to fairly accurately simulate the flow in the near-border layer. Results of numerous studies indicate that the model of turbulent viscosity SST more accurately describes the flow in the near-border layer than the model of turbulence k–ε.

Vector velocity field for the fine adaptive grid for two models of turbulence (Fig. 6) illustrates a difference in the flow-over character. The tear point of the near-border layer on the blades of guiding device when calculating using the model SST is higher, one can also spot the difference in the character of flow-over in the trace after the blades of the guiding device. However, calculations that were performed using the model of turbulent viscosity SST required longer computing time. Thus, at the first phase of designing, in order to save computing time, we can recommend using the model of turbulent viscosity k–ε and the coarse grid. Further, in a more detailed study of losses and character of the flow in the near-border layer and aerodynamic trail, it is necessary to employ the model of turbulent viscosity SST and the fine adaptive grid.

Results obtained in the present work are planned to be applied when solving the problem of tearing flow in the stages of axial compressors and fans.

## 7. Conclusions

1. The applied 3D model of the stage of an axial compressor with different variants of topology of the computational grid allows defining the parameters of flow when using different models of turbulent viscosity. A model of the degree of an axial compressor with the fine adaptive grid makes it possible to explore in detail parameters of the near-border layer.

2. We obtained results of the computational research into the flow using the models of turbulent viscosity SST and k–ε. Comparing the results of numerical and physical experiments revealed that the application of models of turbulent viscosity SST and k–ε provides for determining the parameters of flow with errors of 0.3–8.6 %.

3. Results of the study showed that at the first phase of calculation of the stage of an axial compressor one can recommend using the model of turbulent viscosity k–ε and the coarse computational grid. To solve the problems on internal aerodynamics of compressors taking into account the flow in the near-border layer and aerodynamic trail, it is advisable to employ the model of turbulent viscosity SST and the fine adaptive grid.

## References


1. Tereshhenko, Ju. M. Aerodinamicheskoe sovershenstvovanie lopatochnyh apparatov kompressorov [Text] / Ju. M. Tereshhenko. – Moscow: Mashinostroenie, 1987. – 168 p.
2. Brusilovskiy, I. V. Aerodinamika osevyh ventiliatorov [Text] / I. V. Brusilovskiy. – Moscow: Mashinostroenie, 1984. – 240 p.
3. Pinto, R. N. Computational Fluid Dynamics in Turbomachinery: A Review of State of the Art [Text] / R. N. Pinto, A. Afzal, L. V. D'Souza, Z. Ansari, A. D. M. Samee // Archives of Computational Methods in Engineering. – 2016. doi: 10.1007/s11831-016-9175-2
4. Naseri, A. Numerical investigation of effect of inlet swirl and total-pressure distortion on performance and stability of an axial transonic compressor [Text] / A. Naseri, M. Boroomand, S. Sammak // Journal of Thermal Science. – 2016. – Vol. 25, Issue 6. – P. 501–510. doi: 10.1007/s11630-016-0891-6
5. Tereshhenko, Ju. M. Flow simulation in compressor cascades at high angles of attack [Text] / Ju. M. Tereshhenko, E. V. Doroshenko, Dzh. Abolhassan zade // Eastern-European Journal of Enterprise Technologies. – 2015. – Vol. 4, Issue 8 (76). – P. 26–30. doi: 10.15587/1729-4061.2015.47206
6. Tereshhenko, Ju. M. Influence research of profile chords ratio on aerodynamic characteristics of tandem compressor cascade [Text] / Ju. M. Tereshhenko, E. V. Doroshenko, Dzh. Abolhassan zade // Eastern-European Journal of Enterprise Technologies. – 2015. – Vol. 5, Issue 8 (77). – P. 9–13. doi: 10.15587/1729-4061.2015.50535
7. Song, P. Blade shape optimization for transonic axial flow fan [Text] / P. Song, J. Sun // Journal of Mechanical Science and Technology. – 2015. – Vol. 29, Issue 3. – P. 931–938. doi: 10.1007/s12206-015-0207-x
8. Janke, C. Compressor map computation based on 3D CFD analysis [Text] / C. Janke, D. Bestle, B. Becker // CEAS Aeronautical Journal. – 2015. – Vol. 6, Issue 4. – P. 515–527. doi: 10.1007/s13272-015-0159-y
9. Yin, S. Application and comparison of SST model in numerical simulation of the axial compressors [Text] / S. Yin, D. Jin, X. Gui, F. Zhu // Journal of Thermal Science. – 2010. – Vol. 19, Issue 4. – P. 300–309. doi: 10.1007/s11630-010-0387-8
10. Liu, Y. Modified k-ω model using kinematic vorticity for corner separation in compressor cascades [Text] / Y. Liu, H. Yan, L. Fang, L. Lu, Q. Li, L. Shao // Science China Technological Sciences. – 2016. – Vol. 59, Issue 5. – P. 795–806. doi: 10.1007/s11431-015-6005-y
11. Krivosheev, I. A. Choice model of turbulence in calculating loss of pressure in the flow of the gas turbine engine using software ANSYS CFX [Text] / I. A. Krivosheev, A. Ju. Chechulin, Ju. A. Hohlova // Vestnik UGATU. – 2011. – Vol. 15, Issue 2 (42). – P. 68–73.

12. Bykov, L. V. Vychislitel'naya mehanika sploshnyh sred v zadachah aviacionnoy i kosmicheskoy tehniki [Text] / L. V. Bykov, A. M. Molchanov, M. A. Shherbakov, D. S. Janyshev. – Moscow: LENAND, 2015. – 668 p.
13. Menter, F. R. Two-equation eddy-viscosity turbulence models for engineering applications [Text] / F. R. Menter // AIAA Journal. – 1994. – Vol. 32, Issue 8. – P. 1598–1605. doi: 10.2514/3.12149
14. Jun, A. A. Teoriya i praktika modelirovaniya turbulentnyh techeniy [Text] / A. A. Jun. – Moscow: Librokom, 2009. – 274 p.
15. Svechnikov, V. S. Aerodinamicheskie harakteristiki stupeni osevogo kompressora [Text] / V. S. Svechnikov, A. B. Kirillov. – Moscow: CAGI, 1958. – 94 p.
16. Boiko, A. V. Primenenie vychislitel'noi aerodinamiki k optimizacii lopatok turbomashin [Text] / A. V. Boiko, Ju. N. Govorushhenko, M. V. Burlaka. – Kharkiv: NTU «KhPI», 2012. – 192 p.

UDC 628.33
DOI: 10.15587/1729-4061.2017.100835

# COMPUTER SIMULATION OF HYDRAULIC FLOW IN A MIXING DEVICE WITH A DIAPHRAGM OF SPECIAL DESIGN INSTALLED IN IT

**A. Shevchenko**
Engineer
Prodeko-Elk Sp.zo.o.
Strefowa str., 9, Elk, Poland, 19-300
E-mail: shevchenko950@gmail.com

**T. Shevchenko**
PhD, Associate Professor
Department of Water Supply, Sanitation and Clean Water
O. M. Beketov National University of Urban Economy in Kharkiv
Marshal Bazhanov str., 17, Kharkiv, Ukraine, 61002
E-mail: Tamara.Shevchenko@kname.edu.ua

***Наведено результати комп'ютерного моделювання гідравлічного потоку в змішувальному пристрої з діафрагмою спеціальної конструкції. За допомогою пакета програмного забезпечення «FlowVision» (м. Москва, Російська Федерація) отримані віртуальні моделі потоку, показані та змодельовані епюри розподілу тиску, швидкостей і турбулізація потоку. Ці фактори зумовлюють активне і повне перемішування розчину реагенту та оброблюваної води***

***Ключові слова: статичний змішувач, комп'ютерне моделювання, епюра розподілу тиску і швидкостей, турбулентна дисипація***

***Приведены результаты компьютерного моделирования гидравлического потока в смесительном устройстве с диафрагмой специальной конструкции. С помощью пакета программного обеспечения «FlowVision» (г. Москва, Российская Федерация) получены виртуальные модели потока, показаны и смоделированы эпюры распределения давления, скоростей и турбулизация потока. Эти факторы обуславливают активное и полное перемешивание раствора реагента и обрабатываемой воды***

***Ключевые слова: статический смеситель, компьютерное моделирование, эпюра распределения давления и скоростей, турбулентная диссипация***

## 1. Introduction

When coagulating water impurities, a rapid and even distribution of reagents in its volume is necessary. This ensures maximum contact of impurity particles with intermediate products of the coagulant hydrolysis. The formed hydrolysis products exist for a short time. Such short period of time is due to the fact that the processes of hydrolysis, polymerization and adsorption occur within 1 s [1, 2].

Perkinetic coagulation ends when the particles reach size of 1...10 μm, which practically coincides with the period of rapid distribution of the coagulant in the water treated in mixers. Inefficient mixing leads to an over-consumption of coagulant and a low agglomeration rate of water impurities at a given dose of reagent. Therefore, it is necessary to create an optimum operating mode of the mixers in which the coagulant would come into contact with a maximum number of particles of water impurities before hydrolysis and polymerization reactions terminate.

To uniformly and quickly mix reagents with water, one should introduce them in the zones of the greatest turbulence of the flow at several points of its cross section. To mix the reagent with water, it is necessary to provide reagent distributors (reagent injection devices) and mixers. These devices ensure a rapid and uniform distribution in the feed channel or pipeline. A subsequent intensive mixing of the reactants with the treated water takes place in the mixers. Reagent distributors are recommended to be made in a form of perforated tubular systems or inserts in the pipeline, which serve as local resistance.

Mixing of reagents with the treated water is carried out in mixing devices (Venturi nozzles, diaphragms), pipe