UDC 621.891 DOI: 10.18372/0370-2197.4(109).20762

V. Ye. BRESHEV, O.V. BRESHEV, P.L. NOSKO, O.V. BASHTA, S.Yu. BOHDAN

State University "Kyiv Aviation Institute", Ukraine

SIMULATION MODELING IN HYDRODYNAMIC ANALYSIS SOFTWARE FOR THE FLOW OF PUMPED FLUID IN A CENTRIFUGAL PUMP IMPELLER

This paper presents the methodology and results of simulation modeling of the flow of pumped fluid within a centrifugal pump impeller, along with a comparison to analytical calculation results. The functional performance of dynamic pumps, including generated head, capacity, axial and radial forces, and efficiency, is directly governed by internal fluid flow parameters. Classical analytical models often rely on simplifying assumptions that significantly diverge from real physical flow behavior due to the theoretical complexity of analyzing 3D, non-uniform (turbulent) flow. The use of modern Computer-Aided Engineering (CAE) tools allows for high-fidelity 3D simulation, enabling researchers to drastically reduce or entirely eliminate the need for physical prototypes and significantly shorten the overall design cycle. A structured methodology for simulation modeling is developed and applied here to analyze liquid flow in the impeller of a novel shaftless centrifugal pump design. Deviations between the outcomes obtained by the two methods do not exceed 15%, which is acceptable for the design of pumps and other turbomachinery. Based on the conducted research, a general scheme for computer-aided design (CAD) of pumps is proposed

Keywords: *simulation modeling, methodology, fluid, thermodynamic parameters, integral characteristics,centrifugal pump impeller.*

Introduction. The development of methodologies—and, in fact, the practical implementation—of simulation modeling of fluid flows using CAD/CAE tools in the design and investigation of machinery is relevant for several reasons.

Firstly, for many types of machines, the functional performance and key characteristics are directly governed by fluid or gas flow parameters during operation. For example, the flow parameters of gaseous (air) lubrication in gas-static bearing clearances determine their load capacity and stiffness, which significantly influence output characteristics of machine drives—such as natural vibration frequencies, allowable speed ranges, permissible axial and radial loads, and efficiency [1–3]. Similarly, flow parameters of the pumped gas in centrifugal compressors define their capacity, compression ratio, and efficiency. In dynamic (centrifugal) pumps, the flow characteristics within the impeller determine the generated head, capacity, axial and radial forces acting on the impeller, hydraulic losses, torque, and power consumption. Consequently, the physical processes of fluid or gas flow, which critically affect machine parameters and performance, must be thoroughly investigated during design or modernization stages.

Secondly, the theoretical complexity of analyzing gas- and hydrodynamic processes in machines—especially when quantifying parameters functionally linked to machine geometry (e.g., the flow passage geometry of a centrifugal impeller or the clearance in gas-lubricated bearings)—presents significant challenges. Due to the three-dimensional spatial distribution of kinematic and thermodynamic (pressure, temperature, density) flow parameters and flow non-uniformity (turbulence), mathematical modeling and derivation of analytical relationships are highly difficult.

Resulting analytical solutions are often approximate, relying on simplifying assumptions that significantly diverge from real physical flow behavior [1, 3–5]. As a result, machine designers and manufacturers are often compelled to conduct numerous physical prototype tests, iteratively refining designs to achieve target performance. A typical example is the development of jet engines: despite well-established theories of fuel combustion in combustion chambers and gas expansion through Laval nozzles, analytical models alone cannot eliminate the need for extensive bench testing of propulsion systems.

Today, the use of modern computational tools and CAE software for gas- and hydrodynamic analysis has fundamentally transformed the study of fluid flows in machinery. Computational experiments based on detailed 3D models of machines or their components enable high-fidelity simulation of liquid and gas flow processes, yielding spatial distributions of key parameters and reliable integral characteristics [6–8]. At the current stage of digital technology development, simulation modeling allows researchers and engineers to:

- ✓ drastically reduce—or in some cases entirely eliminate—the need for physical prototypes and associated experimental measurements;
- ✓ significantly shorten the overall design and research cycle;
- ✓ reduce overall R&D costs.

Problem Statement. Simulation modeling of physical processes—particularly fluid and gas flows—is now one of the most effective approaches both for scientific inquiry and for solving practical engineering design problems in new machinery development. While CAD/CAE engineering analysis tools are rapidly evolving [6–8], direct application of CAE software tools and standard instructions often fails to yield sufficiently reliable results that can be validated by alternative methods. This limitation stems from the large number of available modeling tools, configuration options, and the high variability of initial and boundary conditions. For instance, a simple parametric sweep involving 5 factors each varied at 5 levels produces 3,125 possible values of load capacity for gas-lubricated conical gas-static bearings [1]; with 8 factors at 7 levels each, the number exceeds 5.77 million. Therefore, researchers and design engineers must clearly define the objective, model setup, initial and boundary conditions, align them with the overall research or design plan, perform a priori estimation of expected outcomes, and select appropriate validation methods. Thus, there is a clear need to develop a structured methodology for simulation modeling [9,10]. In this work, such a methodology is developed and demonstrated through the study of liquid flow in the impeller of a novel “shaftless” centrifugal pump design.

Objectives of the Study.

- ✓ Develop a methodology for simulation modeling of fluid flow, exemplified by the analysis of liquid flow in the impeller of a shaftless centrifugal pump;
- ✓ Obtain simulation results showing spatial distributions of velocity and pressure of the pumped fluid within the impeller, followed by processing and analysis;
- ✓ Propose a generalized algorithm for dynamic pump design incorporating simulation-based flow analysis in the impeller.

Research Methodology. The methodology involves defining the overall simulation workflow, specifying requirements for the algorithm and expected results, conducting computational experiments, analyzing outcomes, and integrating findings into a generalized computer-aided pump design framework.

Key stages of simulation preparation and execution include:

1. *A priori analysis of physical processes* during machine operation that govern performance and depend on fluid flow behavior. For pumps, these include: flow rate (capacity), generated head, cavitation-free operation conditions, vibration levels, forces and moments acting on the impeller, thermodynamic parameter distributions along the flow path, torque on the impeller, and power transferred to the fluid.

2. *Development of a 3D model* of the machine component where fluid motion occurs—in this case, a dual centrifugal impeller integrated with a ring-shaped rotor that enables stable rotation without mechanical bearings or a shaft (Fig. 1).

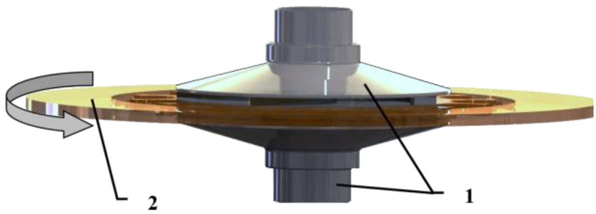

Fig. 1. 3D model of the dual impeller (1) mounted on a ring rotor (2)

The investigated pump features a dual impeller design to counteract axial forces generated by fluid redirection in each individual impeller. The impeller assembly is fixed to a ring rotor that rotates in an electromagnetic field without mechanical support, where electromagnetic forces provide both driving torque and stabilization. Since both impellers are identical, simulation of flow in one is sufficient.

A disassembled 3D view of the shaftless centrifugal pump is shown in Fig. 2.

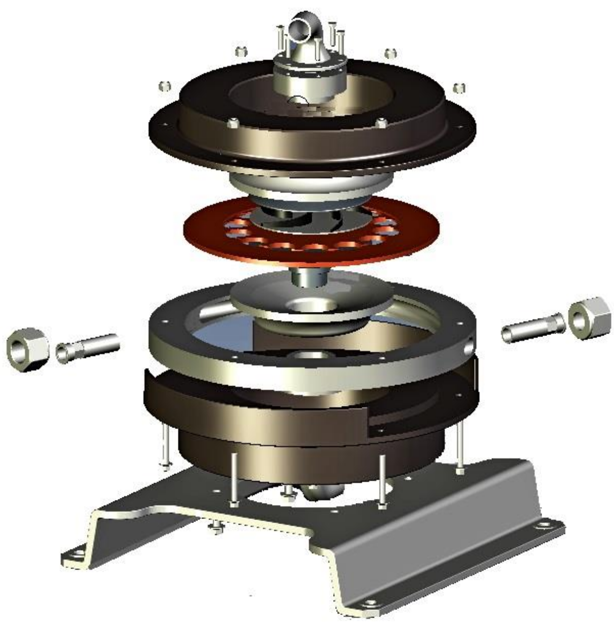

Fig. 2. Exploded 3D model of the pump

3. *Definition of core requirements* for the simulation algorithm and results:

- ✓ Deviation of integral characteristics (e.g., head, flow rate, torque) from analytical or experimental reference values must not exceed 15%;
- ✓ Sensitivity to changes in inlet pressure must be at least 0.01 atm;
- ✓ Sensitivity to flow rate variations must be at least 0.01 L/min;
- ✓ Convergence across repeated simulations must be ≥99%.

4. *Creation of the computational domain* (impeller geometry) and specification of initial and boundary conditions in CFD software. The 3D impeller model is modified by adding virtual solid "caps" at the inlet and outlet (Fig. 3). These caps define the fluid volume and serve as surfaces for applying boundary conditions, while remaining non-intrusive to the original CAD geometry.

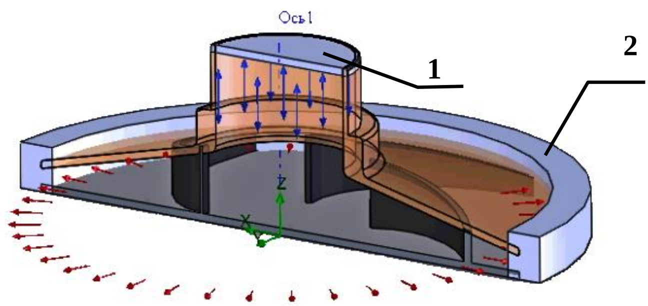

Fig. 3. Cross-sectional view of the impeller with inlet (1) and outlet (2) caps

In the CFD setup:

- ✓ A convenient unit system is selected;
- ✓ Internal flow analysis type is chosen;
- ✓ Direction of gravity is specified;
- ✓ The impeller is aligned with the global coordinate system and assigned rotational motion (e.g., about the Z-axis at 800 or 1420 rpm);
- ✓ Fluid properties (e.g., water: known density and viscosity) and flow regime (laminar/turbulent, with appropriate turbulence model) are defined;
- ✓ Inlet boundary: pressure = 101,325 Pa, temperature = 20°C;
- ✓ Outlet boundary: volumetric flow rate = 4 L/s (alternatively, pressure could be specified, with flow rate computed as a result);
- ✓ Simulation accuracy (mesh resolution) is set, corresponding to the number of finite control volumes used to discretize the fluid domain.

5. *Execution of simulation:* typically, a series of computational runs is performed to refine model settings and optimize boundary conditions.

6. *Result extraction, visualization, statistical processing (for repeated runs), analysis, and validation* against classical analytical methods or experimental data.

Results and Discussion. Modern mechanical CAD/CAE systems enable visualization of spatial distributions of thermodynamic fluid parameters in multiple formats: numerical arrays, tables, 2D/3D plots, or color-mapped distributions over surfaces or volumes. Users can probe specific points, lines, planes, or surfaces of interest.

Figure 4 shows the velocity distribution of pumped water in the rotating impeller, evaluated in a plane parallel to the impeller base and located at mid-height of the outlet.

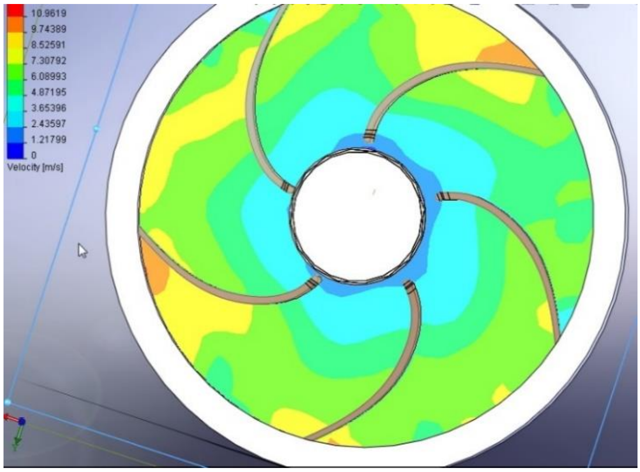

Fig. 4. Velocity distribution of pumped water in the impeller

Simulations were performed at 800 rpm. The fluid domain (bounded by caps and impeller geometry) was discretized into 1,521 control volumes. Thermodynamic flow parameters were computed at the center of each volume. A total of 359 flow trajectories were tracked over 70 iterations per simulation to achieve target convergence.

Figure 5 displays the corresponding pressure distribution in the same plane. Under no-downstream-resistance conditions, the generated head at the impeller outlet reached 1.2 atm.

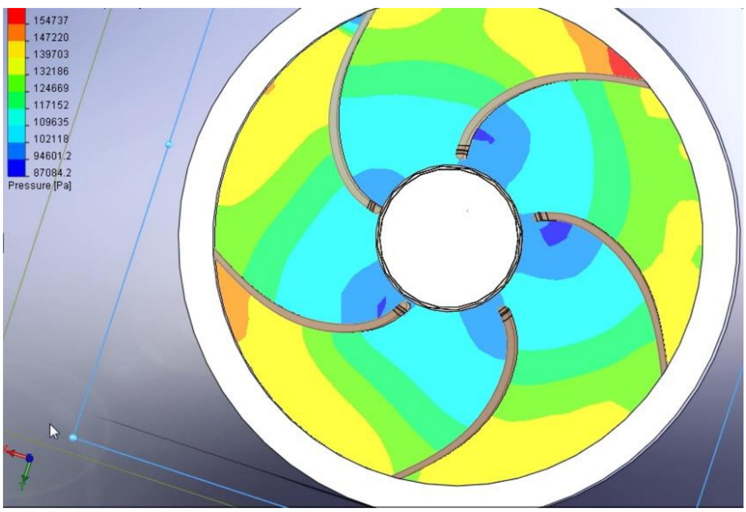

Fig. 5. Pressure distribution of pumped water in the impeller

Figure 6 illustrates 3D streamlines showing fluid trajectories through the rotating impeller, with color indicating local fluid velocity.

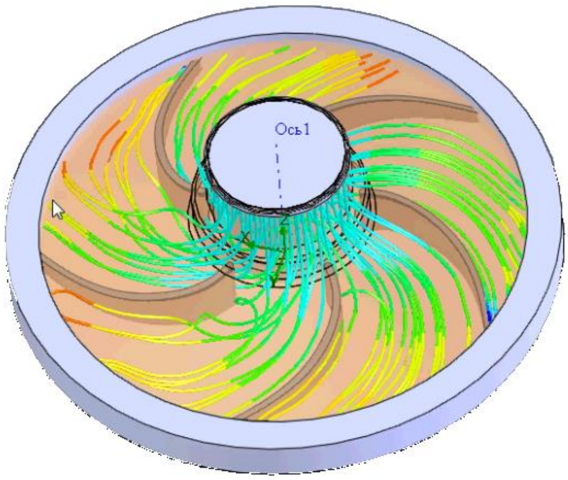

Fig. 6. 3D streamlines of fluid flow in the rotating impeller

Analysis of simulation results confirmed that the obtained flow parameter distributions align with established fluid dynamics principles and classical analytical methods [11]. Comparison of key integral characteristics—specifically:

- ✓ Absolute fluid velocity at impeller inlet (V1) and outlet (V2);
- ✓ Head (H), accounting for hydraulic losses;
- ✓ Pump capacity (Q);
- ✓ Torque on the impeller (M);
- ✓ Power transferred to the fluid (P)—

revealed deviations from analytical calculations of no more than 15%.

Conclusions

1. A general methodology for simulation modeling of fluid flows has been developed, applicable to pumps and other machines where liquid or gas flow governs operational behavior. It includes:

- ✓ A priori physical process analysis;
- ✓ 3D component modeling;
- ✓ Specification of algorithmic and result accuracy requirements;
- ✓ Computational domain setup with defined initial/boundary conditions;
- ✓ Execution of simulation;
- ✓ Result extraction, validation against analytical or experimental data, and engineering interpretation.

2. Simulation of water flow in a centrifugal pump impeller at 800 rpm yielded detailed spatial distributions of velocity and pressure and enabled calculation of integral performance characteristics (flow rate and head). Results aligned with theoretical expectations and deviated by ≤15% from classical analytical predictions.

3. A generalized automated pump design framework has been proposed, with CAE-based simulation modeling at its core, enabling in-depth investigation of fluid flow behavior, thermodynamic parameter distributions, and their functional dependencies on geometry and boundary conditions.

References

1. Breshev, V. E. (2016). Development of Theory and Methods of Design of Contactless Drives with Combined and Passive Stability Assurance. Lugansk: Luhansk Taras Shevchenko National University Publishing House.

2. Rowe, W. B. (2012). Hydrostatic, Aerostatic, and Hybrid Bearing Design. Oxford: Butterworth-Heinemann.

3. Al-Bender, F. (2021). Air Bearings: Theory, Design and Applications. John Wiley & Sons.

4. Kosmynin, A. V. (2002). Gas Bearings of High-Speed Turbo Drives of Metalworking Equipment. Dal'nauka.

5. Pinegin, S. V., Tabachnikov, Yu. B., & Sipenkov, I. E. (1982). Static and Dynamic Characteristics of Gas-Static Supports. Nauka.

6. Khryts'kyi, A. A. (2016). Fundamentals of Design Subsystem Development Based on SOLIDWORKS APS. Kryvyi Rih: KNU Publishing.

7. Howard, W., & Musto, J. (2021). Introduction to Solid Modeling Using SOLIDWORKS 2021. McGraw-Hill Education.

8. Spens, M. (2019). Automating SOLIDWORKS 2019 Using Macros. SDC Publications.

9. Breshev, V. E., & Dolzhenko, Yu. S. (2024). Computational Computer Experiments to Study the Characteristics of the Gas-Static Bearing of the Spindle of a Grinding Machine. Vestnik of Lugansk State University named after Vladimir Dahl, 2(80), 28–35.

10. Kharzhevskyi, V. O. (2025). Automation of Engineering Calculations in Mechanical Engineering. Khmelnytskyi: KhNU.

11. Carravetta, A., Derakhshan Houreh, S. (2018). Pumps as Turbines: Fundamentals and Applications. Springer Tracts in Mechanical Engineering.

Стаття надійшла до редакції 24.11.2025

Breshev Volodymyr Yevhenovych – Doctor of Technical Sciences, Professor, researcher in mechanical engineering, tel. 406-78-42, E-mail: nau12@ukr.net

Breshev Oleksii Volodymyrovych - PhD in Engineering, degree applicant, Department of Applied Mechanics and Materials Engineering, State University "Kyiv Aviation Institute", 1, Lubomyr Huzar Ave. 1, Kyiv, Ukraine, 03058, E-mail: abreshev@gmail.com, https://orcid.org/0009-0007-4176-775X.

Nosko Pavlo Leonidovych – Doctor of Technical Sciences, Professor, Professor of the Department of Applied Mechanics and Materials Engineering, State University "Kyiv Aviation Institute", 1, Lubomyr Huzar Ave., Kyiv, Ukraine, 03058, tel. 406-78-42, E-mail: nau12@ukr.net, https://orcid.org/0000-0003-4792-6460.

Bashta Oleksandr Vasylovych - PhD in Engineering, Associate Professor, Department of Applied Mechanics and Materials Engineering, State University "Kyiv Aviation Institute", 1, Lubomyr Huzar Ave., Kyiv, Ukraine, 03058, E-mail: oleksandr.bashta@npp.kai.edu.ua, https://orcid.org/0000-0001-7914-897X.

Bohdan Svitlana Yuriivna - PhD in Engineering, Associate Professor, Department of Applied Mechanics and Materials Engineering, State University "Kyiv Aviation Institute", 1, Lubomyr Huzar Ave., Kyiv, Ukraine, 03058, E-mail: svitlana.bohdan@npp.kai.edu.ua, https://orcid.org/0000-0002-9506-6357.

В.Є. БРЕШЕВ, О.В. БРЕШЕВ, П.Л. НОСКО, О.В. БАШТА, С.Ю. БОГДАН

МОДЕЛЮВАННЯ В ПРОГРАМНОМУ ЗАБЕЗПЕЧЕННІ ДЛЯ ГІДРОДИНАМІЧНОГО АНАЛІЗУ ПОТОКУ РІДИНИ В РОБОЧОМУ КОЛЕСІ ЦЕНТРИФУГНОГО НАСОСУ

У цій статті представлено методологію та результати моделювання потоку рідини, яка перекачується, в робочому колесі відцентрового насоса, а також порівняння з результатами аналітичних розрахунків. Функціональні характеристики динамічних насосів, включаючи створюваний напір, продуктивність, осьові та радіальні сили, а також ККД, безпосередньо залежать від внутрішніх параметрів потоку рідини. Класичні аналітичні моделі часто базуються на спрощених припущеннях, які значно відрізняються від реальної фізичної поведінки потоку через теоретичну складність аналізу тривимірного, неоднорідного (турбулентного) потоку. Використання сучасних інструментів комп'ютерного проектування дозволяє проводити високоточне тривимірне моделювання, що дає дослідникам можливість значно зменшити або повністю усунути необхідність у фізичних прототипах і значно скоротити загальний цикл проектування. Структурована методологія моделювання симуляції розроблена і застосовується тут для аналізу потоку рідини в робочому колесі нової конструкції безвального відцентрового насоса. Відхилення між результатами, отриманими двома методами, не перевищують 15%, що є прийнятним для проектування насосів та інших турбомашин. На основі проведених досліджень запропоновано загальну схему комп'ютерного проектування (CAD) насосів.

Ключові слова: моделювання, методологія, рідина, термодинамічні параметри, інтегральні характеристики, робоче колесо відцентрового насоса.

Список літератури

1. Breshev, V. E. (2016). Development of Theory and Methods of Design of Contactless Drives with Combined and Passive Stability Assurance. Lugansk: Luhansk Taras Shevchenko National University Publishing House.
2. Rowe, W. B. (2012). Hydrostatic, Aerostatic, and Hybrid Bearing Design. Oxford: Butterworth-Heinemann.
3. Al-Bender, F. (2021). Air Bearings: Theory, Design and Applications. John Wiley & Sons.
4. Kosmynin, A. V. (2002). Gas Bearings of High-Speed Turbo Drives of Metalworking Equipment. Dal’nauka.
5. Pinegin, S. V., Tabachnikov, Yu. B., & Sipenkov, I. E. (1982). Static and Dynamic Characteristics of Gas-Static Supports. Nauka.
6. Khryts’kyi, A. A. (2016). Fundamentals of Design Subsystem Development Based on SOLIDWORKS APS. Kryvyi Rih: KNU Publishing.
7. Howard, W., & Musto, J. (2021). Introduction to Solid Modeling Using SOLIDWORKS 2021. McGraw-Hill Education.
8. Spens, M. (2019). Automating SOLIDWORKS 2019 Using Macros. SDC Publications.
9. Breshev, V. E., & Dolzhenko, Yu. S. (2024). Computational Computer Experiments to Study the Characteristics of the Gas-Static Bearing of the Spindle of a Grinding Machine. Vestnik of Lugansk State University named after Vladimir Dahl, 2(80), 28–35.
10. Kharzhevskyi, V. O. (2025). Automation of Engineering Calculations in Mechanical Engineering. Khmelnytskyi: KhNU.
11. Carravetta, A., Derakhshan Houreh, S. (2018). Pumps as Turbines: Fundamentals and Applications. Springer Tracts in Mechanical Engineering.

Брешев Володимир Євгенович – доктор технічних наук, професор, дослідник в галузі машинобудування тел. 406-78-42, E-mail: nau12@ukr.net.

Брешев Олексій Володимирович – кандидат технічних наук (PhD in Eng. S.), здобувач, Державний університет "Київський авіаційний інститут", кафедра прикладної механіки та інженерії матеріалів; м. Київ, Україна; e-mail: abreshev@gmail.com, https://orcid.org/0009-0007-4176-775X.

Носко Павло Леонідович – доктор технічних наук, професор, професор кафедри прикладної механіки та інженерії матеріалів, Державний університет «Київський авіаційний інститут», пр-т Любомира Гузара, 1, м. Київ, Україна, 03058, тел. 406-78-42, E-mail: nau12@ukr.net, https://orcid.org/0000-0003-4792-6460.

Башта Олександр Васильович - кандидат технічних наук, доцент, кафедра прикладної механіки та інженерії матеріалів, Державний університет "Київський авіаційний інститут", пр-т Любомира Гузара, 1, м. Київ, Україна, 03058, E-mail: oleksandr.bashta@npp.kai.edu.ua, https://orcid.org/0000-0001-7914-897X.

Богдан Світлана Юріївна - кандидат технічних наук, доцент, кафедра прикладної механіки та інженерії матеріалів, Державний університет "Київський авіаційний інститут", пр-т Любомира Гузара, 1, м. Київ, Україна, 03058, E-mail: svitlana.bohdan@npp.kai.edu.ua, https://orcid.org/0000-0002-9506-6357.