

Numerical comparative analysis of the energy conversion process in a semi-open and closed impeller of a centrifugal pump

Numeryczna analiza porównawcza przebiegu konwersji energii w wirniku półotwartym i zamkniętym pompy odśrodkowej

KONRAD BABUL, ANDRZEJ KOCHMAŃSKI, PALACKSONE VONGXAYALATH, TOMASZ DĘBSKI

DOI: 10.17512/INSTAL.2026.09.01

The article presents a numerical comparative analysis of the energy conversion process in semi-open and closed impellers of a centrifugal pump, whose blade systems were designed for identical operating conditions. Based on CFD simulations, the flow structure, pressure distribution, and velocity fields were analyzed for both design configurations. The results indicate that vortical regions develop in the semi-open impeller, leading to significant energy losses, primarily due to axial clearance. The design of the closed impeller enables higher outlet velocities, greater head, and improved efficiency by reducing hydraulic losses and stabilizing flow conditions. At the nominal flow rate ($Q_n = 500 \text{ dm}^3/\text{min}$), the introduction of an axial clearance of 0,5 mm resulted in an approximately 11% reduction in head and a decrease in hydraulic efficiency of more than 6%. Furthermore, these differences become increasingly pronounced as the flow rate decreases for the analyzed impeller configurations. The study confirms the usefulness of numerical methods for analyzing detailed flow phenomena and demonstrates the significant impact of impeller design on energy conversion efficiency in centrifugal pumps.

Keywords: centrifugal pump, semi-open impeller, closed impeller, numerical analysis, CFD

Artykuł przedstawia numeryczną analizę porównawczą procesu konwersji energii w wirniku półotwartym i zamkniętym pompy odśrodkowej, których układy łopatkowe zostały zaprojektowane dla identycznych warunków pracy. Na podstawie symulacji CFD przeanalizowano strukturę przepływu, rozkład ciśnienia oraz pola prędkości dla obu rozwiązań konstrukcyjnych. Uzyskane wyniki wskazują, że w wirniku półotwartym powstają obszary wirowe, których skutkiem są znaczne straty energii, wynikające głównie z obecności szczeliny osiowej. Konstrukcja wirnika zamkniętego pozwala osiągnąć wyższe prędkości na wylocie, większą wysokość podnoszenia oraz wyższą sprawność dzięki ograniczeniu strat hydraulicznych i bardziej stabilnym warunkom przepływu. W efekcie wprowadzenia szczeliny osiowej o wartości 0,5 mm, przy nominalnym natężeniu przepływu ($Q_n = 500 \text{ dm}^3/\text{min}$), doszło do ok. 11% spadku wysokości podnoszenia oraz ponad 6% spadku sprawności. Jednocześnie zauważono, że im przepływ jest mniejszy, tym te różnice są znacznie większe dla obu typów wirników. Przeprowadzone badania potwierdzają przydatność metod numerycznych w analizie szczegółowych zjawisk przepływowych oraz wykazują istotny wpływ konstrukcji wirnika na efektywność konwersji energii w pompach odśrodkowych.

Słowa kluczowe: pompa odśrodkowa, wirnik półotwarty, wirnik zamknięty, analiza numeryczna, CFD

Introduction

Advances in materials engineering and computational software enable design and operational improvements in newly developed machines and equipment. Additionally, numerical flow simulations significantly reduce the time and costs associated with the design and implementation phases of a product. Input data for equipment design result directly from the equipment's operating conditions and expected performance characteristics.

In pump engineering, alongside universal solutions, highly application-specific designs are also quite common. Pumping water containing solid particles such as grass, algae, or sand requires appropriately designed hydraulic channels in the pump section to ensure maximum operational reliability of the entire unit. For contaminated liquids, various semi-open impeller designs are most commonly used, with their structure directly determined by the properties of the pumped medium. The methodology for shaping the flow channel in the meridional

cross-section is described in considerable detail in the literature. Although most studies date back to the 1960s and 1970s [1-4], the computational algorithm has not been updated and continues to be successfully used in impeller design. Thanks to the widespread availability of computational software – either unavailable or limited at the time when most textbooks on centrifugal pump design were written – it has become possible to automate these algorithms to increase efficiency in the development of engineering designs.

dr inż. Konrad Babul ORCID: 0000-0003-2719-3247, Katedra Inżynierii Konwersji Energii, Wydział Mechaniczno-Energetyczny, Politechnika Wrocławska, Corresponding author: e-mail: konrad.babul@pwr.edu.pl; mgr inż. Andrzej Kochmański ORCID: 0009-0007-5352-2004, Absolwent Politechniki Wrocławskiej, Inspektor urządzeń ciśnieniowych w Urzędzie Dozoru Technicznego (UDT), Departament Energetyki Jądrowej UDT, e-mail: andrzej.kochmanski@udt.gov.pl; mgr inż. Palackson Vongxayalath ORCID: 0009-0002-3632-8836, Wydział Mechaniczno-Energetyczny, Politechnika Wrocławska, e-mail: 270310@student.pwr.edu.pl; inż. Tomasz Dębski ORCID: 0009-0001-8837-7149, Wydział Mechaniczno-Energetyczny, Politechnika Wrocławska, e-mail: tomekd0@o2.pl

Data zgłoszenia: 13.05.2026. Data przyjęcia: 26.08.2026. Data publikacji: 22.09.2026

The available literature lacks detailed algorithms for calculating semi-open impellers that account for all their design features, such as the axial clearance between the impeller and the casing front wall. Nevertheless, current scientific research, based in part on numerical flow simulations, aims to explain the design relationships between the impeller and the pump performance parameters achieved [5-8].

In the available literature on centrifugal pump design, studies focusing on semi-open and closed impellers are typically conducted separately, with emphasis placed on their individual hydraulic performance, flow structures, or application-specific advantages. Direct comparative analyses of these two impeller configurations under identical operating and geometric conditions remain relatively scarce. Most published works address selected aspects, such as the influence of clearance, blade geometry, or multiphase flow behavior, rather than providing a comprehensive evaluation of the energy conversion process in both designs. As a result, there is a noticeable gap in the literature on systematic investigations that would enable a clear and consistent assessment of differences in energy-transfer mechanisms and associated hydraulic losses between semi-open and closed impellers.

Although limited in number, several studies have attempted to address selected aspects of the comparison between semi-open and closed impellers. For instance, Ayad et al. [9] demonstrated that increasing axial clearance in semi-open impellers intensifies leakage flows, resulting in a noticeable reduction in head and overall efficiency, thereby highlighting the direct relationship between geometric features and hydraulic losses. Several studies referenced in [9] provided a comparative CFD-based evaluation of different impeller types, indicating that closed impellers generally achieve higher efficiency. At the same time, semi-open configurations exhibit greater robustness under challenging operating conditions, particularly in multiphase flows. Additionally, studies such as those reported in [10] emphasize differences in internal flow structures, including turbulence intensity and secondary flow development, which further influence performance characteristics.

Recent studies confirm that research on centrifugal pump impellers remains an active and evolving field, particularly regarding semi-open impeller configurations. Current investigations increasingly combine advanced CFD techniques with experimental validation to analyze the effects of blade geometry, tip clearance, erosion mechanisms, and complex flow structures. Mansour and

Thévenin [11] experimentally and numerically investigated the influence of blade twisting and tip clearance on the performance of a centrifugal pump operating under air–water two-phase flow conditions, providing valuable validation data for numerical models. Wang et al. [12] conduct CFD simulations to investigate the influence of particle size on vortex formation and erosion behavior in a semi-open centrifugal pump. In a subsequent study [13], the authors extended this work by analyzing the effect of the particle on vortex–wall interactions and erosion mechanisms. These studies demonstrate that numerical modeling, supported by experimental investigations, has become an indispensable tool for analyzing complex flow phenomena in centrifugal pumps. At the same time, they confirm that studies directly comparing the hydraulic performance and energy conversion mechanisms of closed and semi-open impellers under identical operating conditions remain relatively limited.

Despite these contributions, the literature still lacks a detailed, systematic analysis that explicitly focuses on the mechanisms of energy conversion in both impeller types. In particular, there is a shortage of studies that simultaneously evaluate pressure rise, energy transfer processes, and hydraulic losses under fully comparable conditions. Therefore, the present work aims to address this gap by conducting a numerical comparative analysis using an identical baseline geometry, with only the impeller type (semi-open versus closed) varied, enabling a consistent and physically meaningful assessment of differences in energy conversion mechanisms. A key aspect of the proposed approach is ensuring comparability and stability of operating parameters across both impellers in numerical simulations, which is often difficult to achieve in experimental investigations due to measurement limitations and sensitivity to operating conditions. Furthermore, CFD-based analyses enable detailed examination of local flow phenomena, including vortex structures, secondary flows, and other localized disturbances that may occur within flow channels.

Methodology of numerical CFD analysis

Design and operating parameters

Design calculations based on the algorithm presented in [2], for the assumed operating parameters corresponding to a pump capacity of $Q_n = 500 \text{ dm}^3/\text{min}$ and a head of $H_n = 26 \text{ m}$, were performed using PTC Mathcad and MS Excel. The main impeller dimensions were determined, including the hub diameter, number of blades and their thickness, the widths of the flow channels at

the inlet and outlet, and the impeller inlet and outlet diameters.

In the next stage, the geometry of the flow channel in the meridional plane was defined, followed by determining the impeller blade profile with single curvature using the point-by-point method. Based on the design characteristics obtained, a parametric CAD model of the impeller geometry was developed. Fig. 1 presents an overview of a well-prepared model of a five-bladed semi-open impeller below.

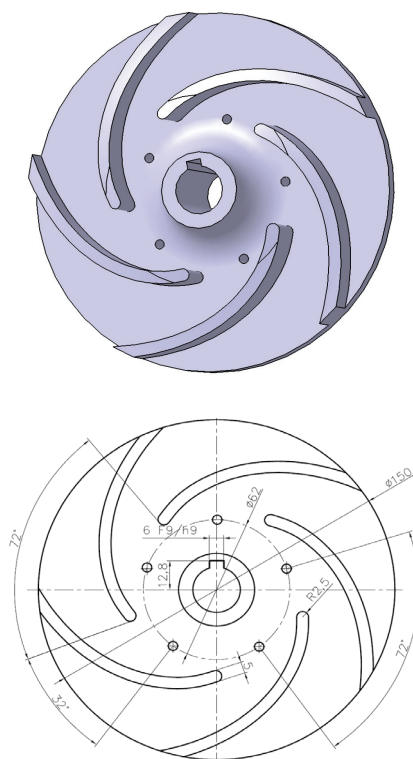


Fig. 1. Model of a semi-open centrifugal impeller with five blades

Rys. 1. Model półotwartego wirnika odśrodkowego z pięcioma łopatkami

Numerical CFD model

Because a centrifugal impeller can be divided into $1/z$ geometrically similar sections with periodic flow patterns, the Ansys BladeGen 2020R2 program was selected to generate the impeller surface model. Reducing the analyzed impeller domain to $1/z$ of its full geometry yields a denser finite-volume mesh and reduces computational time compared to simulations on the full impeller. Fig. 2a presents the surface model of a five-bladed semi-open impeller, while Fig. 2b shows the corresponding sector of the impeller that is used for the numerical calculations.

To perform simulations using the finite volume method, the next step is to discretize the fluid domain within the impeller, as shown in Fig. 3.

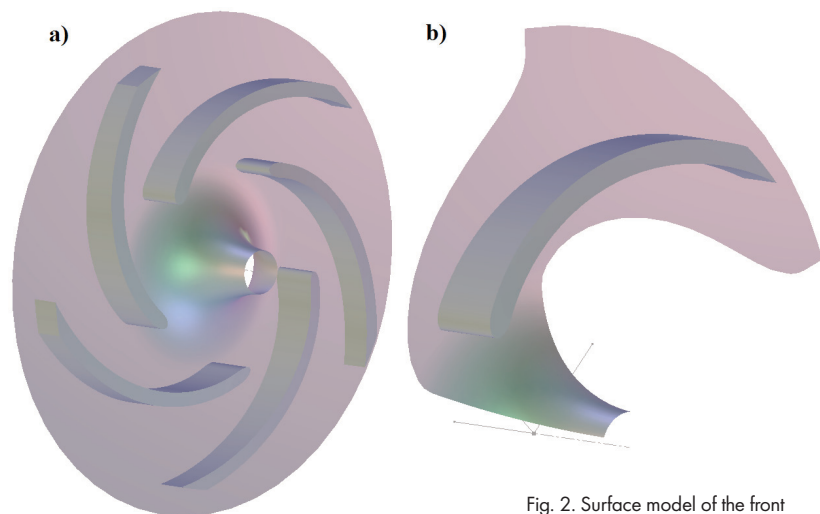


Fig. 2. Surface model of the front geometry of a five-bladed semi-open impeller prepared for: a) the full impeller, b) a 1/3 sector of the impeller

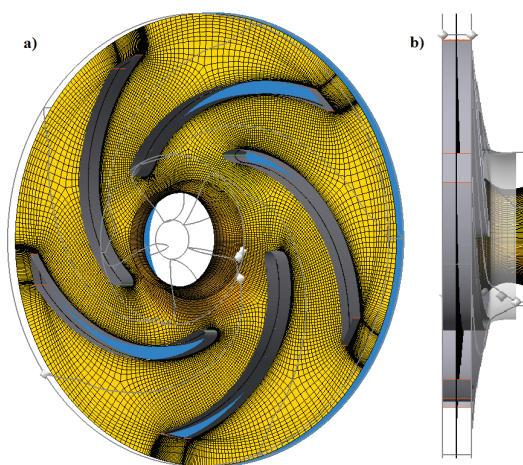


Fig. 3. Surface model of the impeller blades with the finite volume mesh layer applied in the central part of the flow channel: a) front view, b) meridional view

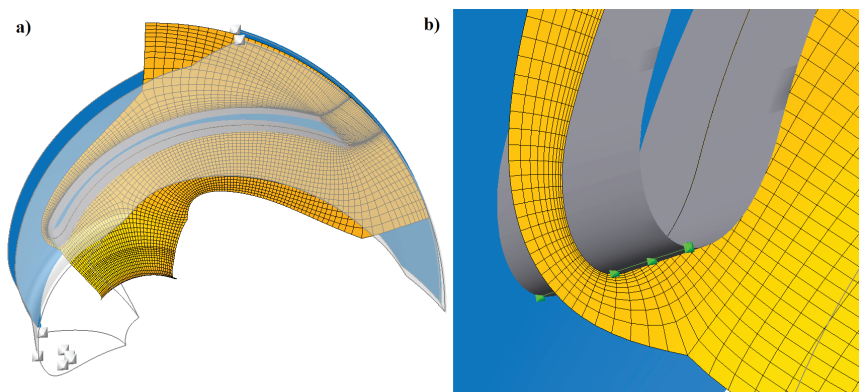


Fig. 4. Surface model of a 1/3 sector of the impeller with the finite volume mesh layer applied in the central part of the flow channel: a) front view, b) enlarged view of the mesh in the blade leading-edge region

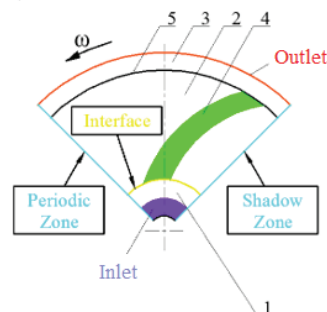


Fig. 5. Schematic representation of the boundary conditions applied to the individual surfaces of the impeller geometric model: a) boundary condition types defined for the 1/3 sector model of the impeller; b) isometric view of the 3D model of the 1/3 sector of the impeller with the defined boundary condition types. 1 - inlet region, 2 - rotating region associated with the impeller blades, 3 - outlet region, 4 - impeller blade, 5 - cylindrical measurement surface

Rys. 5. Schemat nałożenia danego typu warunków brzegowych na poszczególne powierzchnie geometrycznego modelu wirnika: a) zdefiniowane typy warunków brzegowych dla modelu 1/3 części wirnika, b) widok izometryczny na model 3D 1/3 części wirnika ze zdefiniowanymi typami warunków brzegowych; 1 - obszar wlotowy, 2 - obszar ruchu wymuszonego łopatkami wirnika, 3 - obszar wylotowy, 4 - łopatka wirnika, 5 - cylindryczna powierzchnia pomiarowa

The computational mesh was created using ANSYS TurboGrid 2020R2, a dedicated meshing tool for turbomachinery applications, including centrifugal pumps. Mesh generation parameters were selected to obtain a high-quality structured computational mesh. For greater clarity, only the surface of the mesh that passes through the center of the impeller flow channel along the midstream line is shown. The finite volumes used for discretization are hexahedral. They are refined in regions of high velocity and pressure gradients within the impeller, namely on the blade surface and in the vicinity of the blade trailing edge (Fig. 4).

The considered discrete fluid model was divided into subdomains: the inlet region, the region of flow driven by the impeller blades, and the outlet region. Due to the distinctly different flow characteristics in these areas, they were constructed as separate fluid domains with identical physical properties and then coupled using an interface-type boundary, as shown in Fig. 5. The inlet region was defined as a stationary fluid domain. In contrast, the impeller region was defined as a rotating fluid domain using the Multiple Reference Frame model. Due to the cyclic periodicity of the flow within the pump impeller, periodic boundary conditions were applied to the analyzed 1/3 sector of the impeller.

Since the primary objective of the analysis was to compare the operating characteristics of the closed and semi-open impellers, the following simplifying assumptions were adopted:

- constant water density of $998,2 \text{ kg/m}^3$,
- constant dynamic viscosity of $0,001003 \text{ kg/(m}\cdot\text{s)}$,
- the energy equation was not solved; therefore, the effects of water temperature variations were neglected.

Based on the simplifying assumptions presented above, viscous dissipation was neglected because the objective of the study was to compare the hydraulic performance of the closed and semi-open impellers in terms of momentum transfer and turbulence effects.

The numerical simulation of the fluid flow was performed using Ansys Fluent 2020R2. Defining a given process in CFD software involves selecting appropriate mathematical models that describe the phenomena under consideration, followed analogously to solving differential equations, by specifying boundary conditions. In the calculations of the semi-open and closed centrifugal impeller, the k- ϵ turbulence model (Realizable, Enhanced Wall Treatment) was applied with default settings (Model constants: C2-Epsilon = 1,9; TKE Prandtl Number = 1; TDR Prandtl Number = 1,2). According to the ANSYS documentation [14, 15], the Enhanced Wall Treatment is recommended for the k- ϵ turbulence model and is particularly suitable for flows with strong pressure gradients, flow separation, flows involving rotating walls, and highly three-dimensional near-wall flow, which are characteristic features of centrifugal pump impellers. A similar approach employing the k- ϵ turbulence model was adopted in [16], where the model was shown to provide good agreement with experimental results and accurately predict the flow field in a centrifugal pump.

The solution process for a fully defined steady-state problem is carried out by prescribing a certain number of iterations. There is no universal criterion for assessing solution convergence. For some classes of problems, monitoring residuals (imbalances) after each computational iteration is helpful; however, for other types of problems, this approach may lead to misleading conclusions. Therefore, good practice is to assess convergence not only based on residuals from successive iterations but also on appropriately integrated quantities and by verifying mass and energy balance.

Numerical flow simulations, with boundary conditions corresponding to nominal operating parameters ($Q_n = 500 \text{ dm}^3/\text{min}$, $n_n = 3060 \text{ rpm}$), were performed for a five-bladed semi-open impeller with an axial clearance of 0,5 mm as well as for a closed impeller. As a result of 500 computational iterations, convergence of all variables (continuity, momentum) was achieved at a level significantly lower than the default threshold value of 10^{-3} .

Additionally, convergence was verified using the volumetric flow rate and the average pressure and velocity at the impeller inlet and outlet, which are key variables characterizing the flow. Analysis of the variation of these parameters over successive iterations shows (Fig. 6) that after approximately 220 iterations, their values stabilize, indicating convergence of the solution. A grid independence study was also conducted, confirming the reliability and accuracy of the obtained results. The average

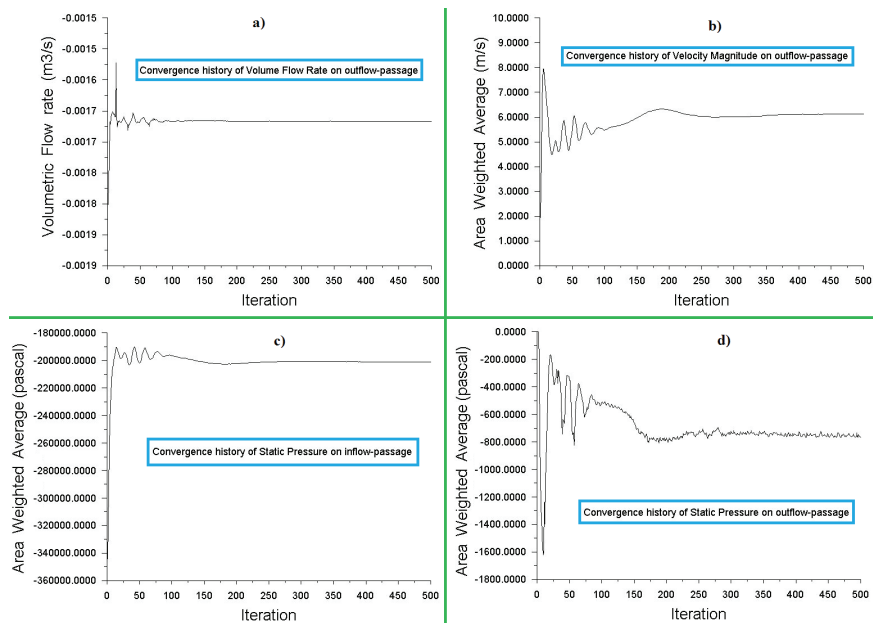


Fig. 6. Characteristics of the variation of control quantities over successive computational iterations performed for a five-bladed semi-open impeller with an axial clearance of 0,5 mm. The individual plots show the variation of the following parameters: a) volumetric flow rate at the impeller outlet, b) mean velocity at the impeller outlet, c) mean static pressure at the inlet to the blade-driven flow region, and d) mean static pressure at the impeller outlet

Rys. 6. Charakterystyki zmian wielkości kontrolnych na drodze kolejnych iteracji obliczeniowych wykonanych dla półotwartego wirnika pięciopłatkowego o wielkości szczeliny 0,5 mm. Poszczególne wykresy przedstawiają zmianę następujących parametrów: a) strumienia objętości na wylocie z wirnika, b) prędkości średniej na wylocie z wirnika, c) średniego ciśnienia statycznego na wlocie do obszaru ruchu wymuszonego łopatkami wirnika, d) średniego ciśnienia statycznego na wylocie z wirnika

pump head at the outlet was selected as the verification parameter, and the mesh independence study was performed using adaptive mesh refinement based on the pressure gradient. The mesh was iteratively refined in regions with the highest pressure gradients until the variation in the predicted pump head between two consecutive refinements was below 2%. For the analyzed 1/z sector of the closed impeller, the final computational mesh consisted of 167380 cells and 185589 nodes. For the semi-open impeller, the adopted mesh-independence criterion was satisfied using a computational mesh with 358895 cells and 387500 nodes.

Analysis and discussion of results

Flow simulation results for a semi-open impeller

Based on the numerical calculations performed for a five-bladed semi-open impeller with an axial clearance of 0,5 mm, the streamline pattern shown in Fig. 7 was obtained. Analysis of the streamline trajectories along the impeller flow channel indicates that they are arranged within the interblade passages in accordance with the impeller's imposed rotational motion. Additionally, as a result of the flow occurring in the clearance between the impeller and the front wall of the casing, the circumferential motion of

the fluid is disturbed. This is reflected in the streamline patterns and ultimately leads to fluctuating flow behavior within the interblade region.

The distribution of static pressure (Fig. 8) was determined at a distance of 5 mm from the impeller shroud. In the inlet region, a zone of reduced pressure can be observed. As the fluid flows through the interblade channels, static pressure increases in stages as the impeller blades transfer momentum to successive fluid layers.

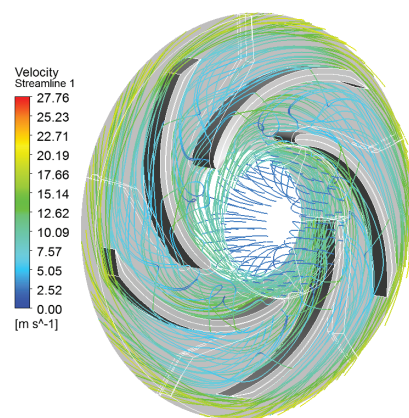


Fig. 7. Streamline pattern obtained from numerical simulations for a five-bladed semi-open impeller with an axial clearance of 0,5 mm
Rys. 7. Przebieg linii prądu wyznaczony na podstawie przeprowadzonych symulacji numerycznych dla półotwartego wirnika pięciopłatkowego o wielkości szczeliny 0,5 mm

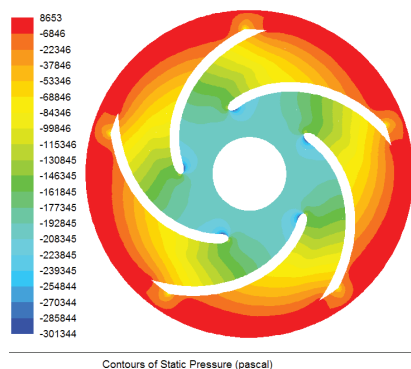


Fig. 8. Static pressure distribution profile in a five-bladed semi-open impeller
Rys. 8. Profil rozkładu ciśnienia statycznego w półotwartym wirniku pięciolopatkowym

The maximum static pressure values obtained directly at the blade outlet correspond to the head generated by the real impeller. Note that the static pressure values shown in the legend on the left are predominantly negative across most of the range. This is because, during the solution of the governing equations, static pressure is referenced to barometric pressure, which is specified at the impeller outlet rather than at the inlet.

The profiles of absolute velocity determined directly at the blade outlet (Fig. 9a) fall within a range that approximately corresponds to the results obtained from the design calculations, for which the absolute velocity at the impeller outlet was 13,9 m/s. Fig. 9b presents the profiles of absolute velocity on the blade surfaces, whose distribution along the impeller radius reflects the increase in linear velocity due to rotational motion at constant angular velocity.

Comparative analysis of the energy conversion in semi-open and closed impellers

Based on the streamline patterns shown in Fig. 10 for both impellers, significant fluctuations occur in the interblade region of the semi-open impeller. These are caused by turbulent disturbances in the boundary layer located near the stationary front wall of the casing. As the impeller blades move in proximity to the casing wall, the fluid boundary layer is periodically disrupted, leading to large velocity gradients between adjacent fluid layers. This, in turn, results in fluctuating disturbances in the streamline patterns.

The profiles of turbulent kinetic energy (Fig. 11) were obtained using the $k-\varepsilon$ turbulence model applied in the calculations. Analysis of the contour distribution of turbulent kinetic energy in the central part of the flow channel of the semi-open impeller clearly reveals regions of very intense turbulent disturbances. These occur immediately

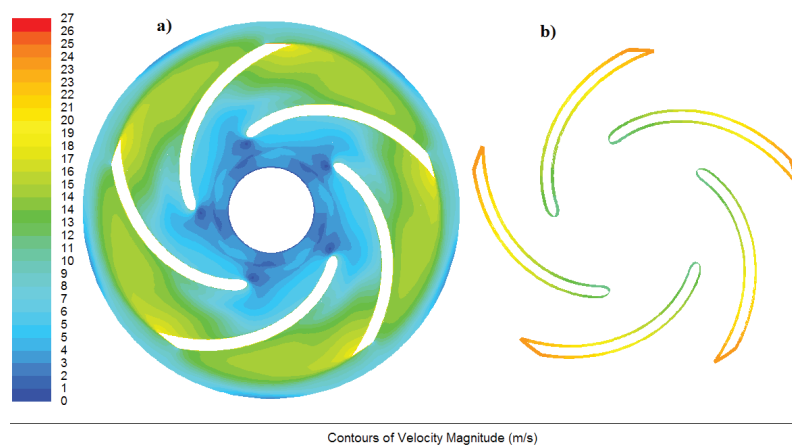


Fig. 9. Velocity profiles in a five-bladed semi-open impeller determined: a) in the fluid domain, and b) along the blade edges
Rys. 9. Profile prędkości w półotwartym wirniku pięciolopatkowym wyznaczone: a) w obszarze płynu, oraz b) na krawędziach łopatek

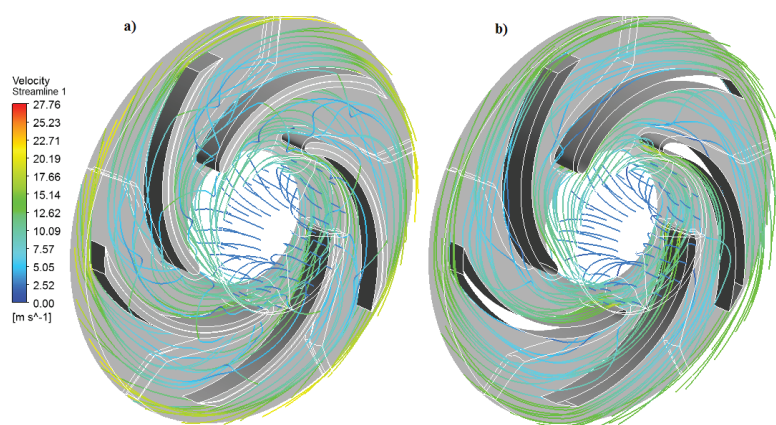


Fig. 10. Streamline patterns obtained from numerical simulations for the impeller: a) a five-bladed semi-open impeller with an axial clearance of 0,5 mm, and b) a closed impeller
Rys. 10. Przebieg linii prądu wyznaczony na podstawie przeprowadzonych symulacji numerycznych dla wirnika: a) półotwartego pięciolopatkowego o wielkości szczeliny 0,5 mm, oraz b) zamkniętego

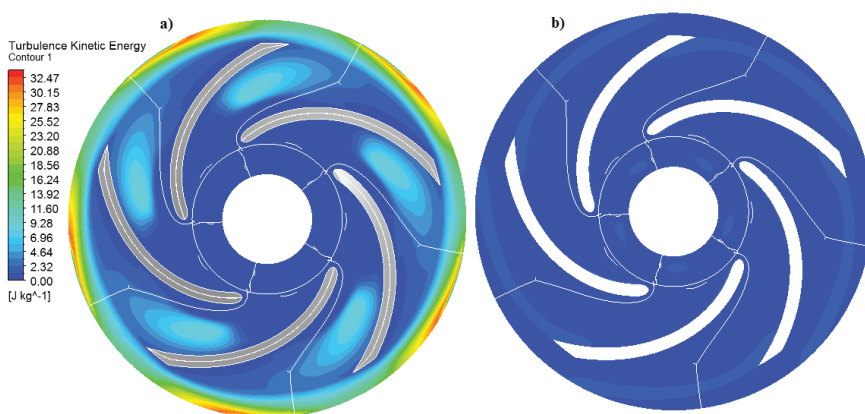


Fig. 11. Profiles of turbulent kinetic energy determined for the impeller: a) a five-bladed semi-open impeller with an axial clearance of 0,5 mm, and b) a closed impeller
Rys. 11. Profile energii kinetycznej turbulencji wyznaczone dla wirnika: a) półotwartego pięciolopatkowego o wielkości szczeliny 0,5 mm oraz b) zamkniętego

downstream of the blade outlet, a phenomenon not observed with the closed impeller. In the pump's hydraulic part, such fluctuations may contribute to energy conversion losses within the volute casing. Additionally, Fig. 11 provides insight into turbulent disturbances that arise within the interblade channel of the semi-open impeller.

Profiles of turbulent kinetic energy (Fig. 12) were obtained on a cylindrical surface that intersects both impellers. In the case of the semi-open impeller, the formation of vortical structures can be observed in the central region of the interblade channel, which does not occur in the case of the closed impeller.

Axial velocity profile in a five-bladed semi-open impeller (Fig. 13), based on which the nature of the occurring vortical structures can be assessed. In turbulent flow developing in the central region of the inter-blade channel, an axial flow component is observed, which may indicate the formation of vortices across the flow passages, as reported by Papierski et al. [5].

In the region located at the impeller blade outlet (Fig. 13), the axial velocity profiles do not exhibit any vortical structures oriented perpendicular to the impeller shroud. This indicates the presence of very intense turbulent flow at the blade outlet of semi-open impellers, which is almost entirely parallel to the impeller shroud. Thus, the velocity vectors of the fluid layers forming vortices in the outlet region have predominantly radial and circumferential components. In contrast, the axial component is relatively small and assumes averaged values comparable to those in other parts of the impeller flow channels. The reason for this behavior may be the formation of a vortex between the front wall of the casing and the tip of the blade side surface. Based on the numerical analyses, these vortical structures reach their maximum intensity at the center of the impeller outlet.

At the outlet of the closed impeller, the velocity is significantly higher than in the case of the semi-open impeller, which is due to losses caused by turbulent flow in the hydraulic passages of the semi-open impeller. Fig. 14 presents the head $H = f(Q)$ and efficiency $\eta = f(Q)$ curves for both impellers, showing that the closed impeller achieves considerably higher head and efficiency. In the presented results, the pump head (H) was determined based on the flow parameters obtained from the CFD simulations, whereas the hydraulic efficiency (η) was calculated according to the following equation:

$$\eta = \frac{P_{\text{hydraulic}}}{P_{\text{motor}}} \quad (1)$$

where: $P_{\text{hydraulic}}$ – hydraulic power determined from the CFD simulation results, W ; P_{motor} – electric motor power determined from the adopted motor performance characteristic, W .

The results of the numerical simulations for the selected design configurations of the closed and semi-open impellers are qualitatively consistent with the experimental investigations and numerical simulations reported in [9] and [5], respectively.

Additionally, the design calculation results were compared with the numerical simulation results obtained for the closed impeller. The application of the design procedure described in [2], implemented in

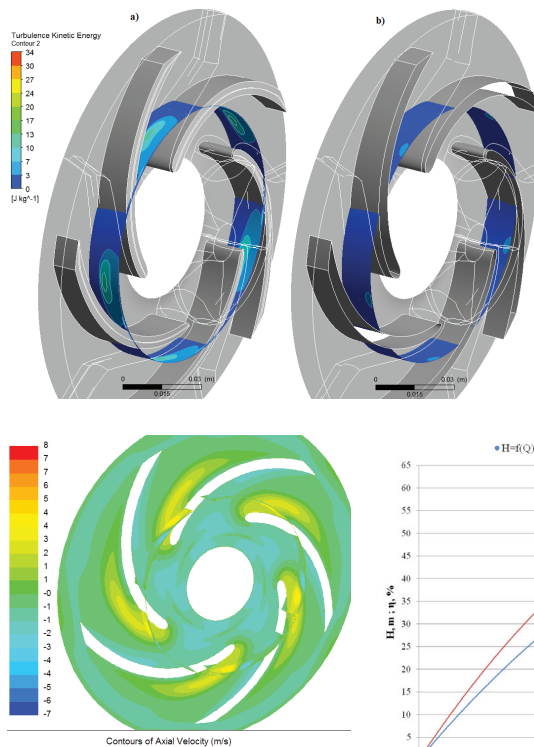


Fig. 12. Profiles of turbulent kinetic energy on a cylindrical surface with a radius of 45 mm, determined for the impeller: a) a five-bladed semi-open impeller with an axial clearance of 0,5 mm, and b) a closed impeller

Rys. 12. Profile energii kinetycznej turbulencji na powierzchni cylindrycznej o promieniu 45 mm wyznaczone dla wirnika: a) półotwartego pięciolopatkowego o wielkości szczeliny 0,5 mm oraz b) zamkniętego

Fig. 13. Axial velocity profiles obtained on a cylindrical surface with a radius of 45 mm and on a plane located 5 mm from the shroud of a five-bladed semi-open impeller with an axial clearance of 0,5 mm

Rys. 13. Profile prędkości osiowej otrzymane na powierzchni cylindrycznej o promieniu 45 mm oraz na płaszczyźnie oddalonej 5 mm od tarczy wirnika półotwartego pięciolopatkowego o wielkości szczeliny 0,5 mm

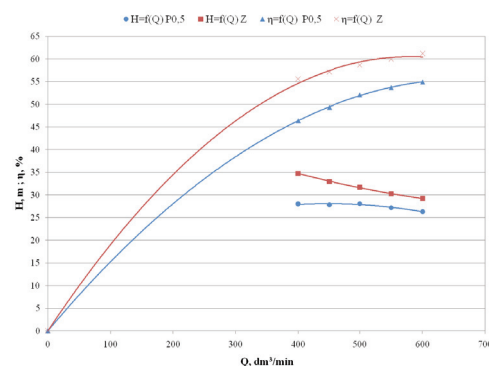


Fig. 14. Performance characteristics $H = f(Q)$ and $\eta = f(Q)$ of a five-bladed closed impeller (Z) and a semi-open impeller with an axial clearance of 0,5 mm (P0,5), determined based on numerical simulations performed in Ansys Fluent

PTC Mathcad and Microsoft Excel, enabled the development of the impeller flow passage geometry, which was subsequently reconstructed in ANSYS BladeGen 2020R2. The calculated pump head (H_{calc}) was 27,5 m, resulting in a design head surplus of $\Delta H = 1,5$ m relative to the assumed nominal head ($H_n = 26$ m). It should be noted, however, that the calculated values correspond to the performance parameters of the complete pump, including, among others, the hydraulic losses occurring in the volute casing. These losses are not accounted for in the numerical model, which is limited to the impeller domain. Consequently, a significantly higher head is expected at the control section directly downstream of the impeller outlet in the numerical model. This observation is consistent with the results presented in Fig. 14.

A quantitative comparison of the numerical simulation results (Fig. 14) indicates that, at the nominal flow rate ($Q_n = 500$ dm³/min), the head and hydraulic efficiency of the closed impeller reached 31,6 m and 58,7%, respectively. In contrast, the corresponding values for the semi-open impeller were 28,1 m and

52,1%. The introduction of an axial clearance of 0,5 mm resulted in an approximately 11% reduction in head and a decrease in hydraulic efficiency of more than 6%. Furthermore, as shown in Fig. 14, these differences become increasingly pronounced as the flow rate decreases for both impeller configurations.

Summary

The influence of impeller design on the energy conversion process was investigated by comparing semi-open and closed impellers under identical operating conditions. Numerical CFD simulations were performed to examine internal flow behavior, including velocity and pressure distributions in the flow channels of both impellers. The analysis revealed that the semi-open impeller is more prone to flow disturbances, particularly due to axial clearance, which promotes leakage and the development of turbulence structures. These effects result in increased energy dissipation and less efficient momentum transfer within the flow passages.

In contrast, the closed impeller ensures a more controlled flow environment, limiting turbulence and reducing hydraulic losses.

Consequently, it achieves better hydraulic performance, reflected in higher head and efficiency values. The numerical simulation results obtained in this study are qualitatively consistent with findings reported in previous studies on this topic. The study also emphasizes the importance of analyzing local flow phenomena, as they significantly influence the machine's global performance. Overall, the results demonstrate that even subtle geometric differences in impeller design can lead to considerable variations in energy conversion efficiency. At the nominal flow rate ($Q_n = 500 \text{ dm}^3/\text{min}$), the introduction of an axial clearance of 0,5 mm resulted in an approximately 11% reduction in head and a decrease in hydraulic efficiency of more than 6%. Furthermore, these differences become increasingly pronounced as the flow rate decreases for the analyzed impeller configurations.

For example, assuming a pump power of 4,4 kW, operation for 16 h per day throughout the year, and a 6% reduction in power demand due to the higher efficiency of the closed impeller, the annual electricity savings would amount to approximately 1,540 kWh. At an electricity price of 1 PLN/kWh, this corresponds to annual savings of approximately 1540 PLN for a single pump. In industrial installations equipped with multiple pumps operating under similar conditions, the cumulative energy and cost savings may therefore become significant.

Future research may investigate the influence of different axial clearances on

performance parameters, particularly in semi-open impellers. Additionally, it would be valuable to analyze the impact of impeller design variations, such as the number of blades, on flow structure, energy losses, and overall pump efficiency. It can also be valuable to analyze the difficulties and costs of fabricating closed impellers compared to open-design impellers in the context of the pump's higher head and overall efficiency.

Acknowledgements

Numerical calculations were performed using resources provided by the Wrocław Center for Networking and Supercomputing (<http://wcss.pl>), grant no. 565.

Literatura

- [1] Łazarkiewicz Sz., Troskolarński T. A., Pompy wirowe, Wydawnictwa Naukowo-Techniczne; Warszawa, 1968
- [2] Jędral W., Pompy wirowe, Wydawnictwo Naukowe PWN, Warszawa, 2001
- [3] Korczak A., Rokita J., Pompy i układy pompowe – obliczenia i projektowanie, Wydawnictwo Politechniki Śląskiej, Gliwice, 1998
- [4] Stępniewski M., Pompy, Wydawnictwa Naukowo-Techniczne, Warszawa, 1978
- [5] Papierski A., Błaszczyk A., Multiobjective optimization of the semi-open impeller in a centrifugal pump by a multilevel method, Journal of Theoretical and Applied Mechanics; Vol. 49, pp. 327-341
- [6] Farid A., El-Azm Aly A. A., Abdalla H. M., Effect of semi-open impeller side clearance on centrifugal pump cavitation inception, International Journal of Computational Physics Series, Vol. 1(2), pp. 24-39, DOI: 10.29167/a1i2p24-39
- [7] Mousmoulis G., Kassanos I., Aggidis G., Anagnostopoulos I., Numerical simulation of the performance of a centrifugal pump with a semi-open impeller under normal and cavitating conditions, Applied Mathematical Modeling, Vol. 89, Part 2, 2021, pp. 1814-1834, DOI: 10.1016/j.apm.2020.08.074
- [8] Zhang F., Heng Y., Liu Q., Tao R., Yang W., Zhu D., Xiao R., Optimization design of a semi-open centrifugal impeller using GA-Fuzzy logic combination strategy, Journal of Applied Fluid Mechanics, Vol. 16, No. 7, 2023, pp. 1427-1441, DOI: 10.47176/jafm.16.07.1655
- [9] Ayad A. F., Abdalla H. M., El-Azm Aly A. A., Effect of semi-open impeller side clearance on the centrifugal pump performance using CFD, Aerospace Science and Technology, Vol. 47, 2015, pp. 247-255, DOI: 10.1016/j.ast.2015.09.033
- [10] Effiom S. O., Abam F., Ofem S., Effiom P.-C., Onochie O., Inah O., Odu P., CFD analysis and hydraulic performance of two-phase flow in a centrifugal pump with rotodynamic multi-impeller configurations, African Journal of Advances in Engineering and Technology, Vol. 1, No. 2, 2025, pp. 5-27
- [11] Mansour M., Thévenin D., Impact of blade twisting and tip clearance on centrifugal pump performance under air–water two-phase flow, International Journal of Heat and Fluid Flow, Vol. 112, 2025, pp. 1-19, DOI: 10.1016/j.ijheatfluidflow.2024.109682
- [12] Wang K., Liu H., Wang L., Guo P., Wang Y., Yang J., Effect of particle size on vortex structure and erosion behavior of semi-open centrifugal pump, Energy, Vol. 293, 2024, pp. 1-20, DOI: 10.1016/j.energy.2024.130576
- [13] Wang K., Jiao Z., Wang Y., Dong J., Liu H., Feng J., Quantitative insights into the effect of particle stokes number on the erosion behavior in semi-open centrifugal pump: Vortex-particle-surface interaction, Powder Technology, Vol. 476, 2026, pp. 1-18, DOI: 10.1016/j.powtec.2026.122360
- [14] ANSYS, Inc., ANSYS Fluent User's Guide, Canonsburg, 2020
- [15] ANSYS, Inc., ANSYS Fluent Theory Guide, Canonsburg, 2020
- [16] Babayigit O., Ozgoren M., Aksoy M. H., Kocacaslan O., Experimental and CFD investigation of a multistage centrifugal pump including leakages and balance holes, Desalination and Water Treatment, Vol. 67, 2017, pp. 28-40, DOI: 10.5004/dwt.2017.20153

KONFERENCJA

MIKROZANIECZYSZCZENIA W ŚRODOWISKU CZŁOWIEKA

POŁĄCZENIE NAUKI
DLA CZYSTSZEJ PRZYSZŁOŚCI

30 LAT
1997-2027

GALA
Z OKAZJI 30-LECIA
ISTNIENIA

WYDZIAŁU INFRASTRUKTURY I ŚRODOWISKA



NAUKA



ŚRODOWISKO



INFRASTRUKTURA



WSPÓŁPRACA

WIEDZA • INNOWACJE • ODPOWIEDZIALNOŚĆ

14 – 16 kwiecień 2027 r., Częstochowa