Conference on Modelling Fluid Flow (CMFF'25)
The 19th International Conference on Fluid Flow Technologies
Budapest, Hungary, August 26 - August 29, 2025

EXPERIMENTAL AND NUMERICAL INVESTIGATION OF THE TURBULENT SWIRLING FLOW IN PIPE BEHIND THE AXIAL FAN IMPELLER

Milan BULAJIĆ, Novica JANKOVIĆ, Lazar LEČIĆ

[1] Corresponding Author. Department for Hydraulic Machinery and Energy Systems, Faculty of Mechanical Engineering, University of Belgrade. Kraljice Marije 16, 11000 Belgrade, Serbia. Tel.: +381 60 585 20 04, E-mail: mbulajic@mas.bg.ac.rs

ABSTRACT (STYLE: ABSTRACT TITLE)

Experimental and numerical research of the turbulent swirling flow in pipe behind the axial fan impeller, which generates solid body profile of the circumferential velocity, is presented in this paper. This is a phenomenon relevant to numerous industrial applications, where axial fans are still inbuilt without the guide vanes. The experimental investigation was carried out using stereo PIV, LDA, and original classical probes. However, in this research are discussed three velocity components measured subsequently by use of the one-component LDA system.

The applied numerical approach involved solving the Navier-Stokes equations under appropriate turbulence modeling conditions, ensuring a high level of accuracy in predicting the swirl intensity and velocity distribution in the pipe. The combination of the unstructured mesh in the inlet and the runner section and structured mesh in the outlet section with the application of the SST and k-ε turbulence models led to the final results. Average velocity profiles, determined in this numerical simulation, are compared with the experimentally obtained data and differences are quantified. These experimentally validated numerical results enable good physical interpretation of the development of the average velocity profiles of the generated turbulent swirling flow in the entire domain.

Keywords: axial fans, CFD, swirling flow, turbulence

Nomenclature

D	[mm]	inner diameter of the pipe
R	[mm]	radial coordinate in the polar cylindrical coordinate system
w	[m/s]	circumferential component of velocity
u	[m/s]	axial component of velocity
z	[mm]	axial coordinate in the polar cylindrical coordinate system
ε	[%]	relative difference between experimental and simulated results

Subscripts and Superscripts

1	section 1 plane
2	section 2 plane

1. INTRODUCTION

This paper presents a computational fluid dynamics (CFD) analysis of turbulent swirl flow generated behind the impeller of an axial fan operating without guide vanes at the outlet. The presence of swirl flow in industrial ventilation and fluid transport systems can significantly impact the overall performance of the system by increasing pressure losses within the pipeline. This, in turn, leads to higher energy consumption and a reduction in system efficiency. Due to these challenges, special attention must be given to the design of axial fan blades when the fan operates without outlet guide vanes, with the goal of minimizing vortex formation and turbulence in the downstream sections. An optimized blade design can help suppress unwanted swirl effects, improving the stability and efficiency of the fluid flow. [1-4]

Previous experimental studies have confirmed the development of swirl flow in the pipelines at the fan outlet. The persistence of this swirl can negatively affect downstream components, leading to vibrations, noise, and potential mechanical fatigue in the ducting system. Understanding the behavior of such flows is essential for optimizing fan performance in various industrial applications, including HVAC systems, cooling systems, and aeration processes. The numerical investigation presented in this paper aims to complement experimental findings by providing detailed insight into the velocity field, pressure distribution, and turbulence characteristics of the swirling flow.

The investigated axial fan was tested according to the standard ISO 5801, case B, which features a free inlet and a ducted outlet configuration. The fan in question is the W30 model, which corresponds to the industrial fan model AP 400, manufactured by Minel, Serbia. It consists of seven blades with an outer diameter of 0.397 m. One of the notable features of this fan is its ability to adjust the blade angle, allowing for variations in flow characteristics based on operational requirements. In this study, the blade angle is set to 30° at the outer diameter to simulate a typical industrial setup. The outlet duct consists of a straight pipe with a length of $27.74 \cdot D$, where the duct diameter D is rounded to 400 mm. This setup ensures that the development of the swirling flow can be accurately captured in both the experimental and numerical analyses.

The experimental investigation was conducted at the Laboratory for Hydraulic Machinery and Energy Systems at the University of Belgrade – Faculty of Mechanical Engineering. A dedicated test rig was constructed to measure flow characteristics using a one-component laser Doppler anemometry (LDA) system. This advanced optical measurement technique provides high-resolution velocity data, which is critical for validating CFD simulations. The experimental setup included two measurement sections placed at z/D = 3.35 and z/D = 26.31 downstream of the fan outlet, ensuring a comprehensive assessment of flow evolution over a significant length of the duct. [1-3]

The fan was tested across a range of rotational speeds, allowing for an evaluation of its performance under different operating conditions. However, in this study, numerical and experimental results are compared specifically for a rotational speed of 2500 rpm, which corresponds to a Reynolds number of Re = 423,350. This flow regime falls within the range of highly turbulent conditions, making it an excellent test case for assessing turbulence models and swirl flow behavior in CFD simulations. [1]

The numerical simulation in this work aims to replicate the experimental conditions and provide a detailed analysis of the flow field, including velocity profiles, pressure variations, and turbulence intensity. The results obtained through CFD are compared with experimental measurements to evaluate the accuracy of the computational approach and its potential application in optimizing axial fan designs for industrial use.

2. EXPERIMENTAL RESEARCH AND FAN IMPELLER CAD PREPARATIONS

The test rig is presented in Figure 1, and is explained in detail in [1]. Measurements were conducted using LDA 1 component system Flow Explorer Mini LDA from Dantec, with smoke fluid EFOG, Density Fluid, Invision, which was freely sucked in the installation. Velocity components were measured subsequently.

A necessary step in the analysis was the 3D scanning of the impeller, allowing for an accurate digital representation to be transferred into CAD software for further modifications and simulations. This process ensured that the geometry of the impeller was precisely captured, enabling a more realistic computational study.

During the CAD modeling phase, certain non-essential components were intentionally excluded to simplify the model without compromising the accuracy of the results. Elements such as the shaft and electromotor were omitted, as their inclusion would have increased the complexity of the numerical analysis while providing little to no improvement in the precision of the simulation. By focusing solely on the impeller and the main flow path, the computational model remained efficient and manageable while still capturing the key aerodynamic characteristics of the fan.

Additionally, the exact geometry of the outlet duct was not fully replicated in the CAD model. Instead, it was assumed that the duct was a perfect cylindrical tube with a diameter of 400 mm. However, in reality, the physical duct contained surface imperfections, creases, and minor irregularities that could influence the flow characteristics. These discrepancies between the idealized model and the actual experimental setup may contribute to differences between the numerical and measured results. Such deviations are common in CFD studies and must be considered when analyzing the accuracy of the computational predictions.

By leveraging 3D scanning technology and CAD modeling, the study aimed to create a well-optimized numerical model that balances accuracy and computational efficiency. The assumptions made in the modeling process are carefully evaluated when comparing the simulation results with experimental data, ensuring a thorough understanding of the potential sources of error.

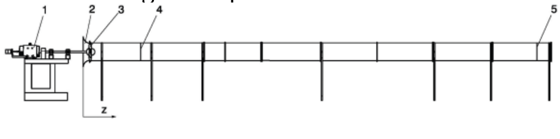

Figure 1 Experimental test rig: 1 - DC motor with electrical power 5 kW, 2 – profiled free bell-mouth inlet, 3- axial fan (swirl generator), 4 – measuring section 1, 5 – measuring section 3. [1]

Figure 2 Axial fan (W30), outside of its test rig [1]

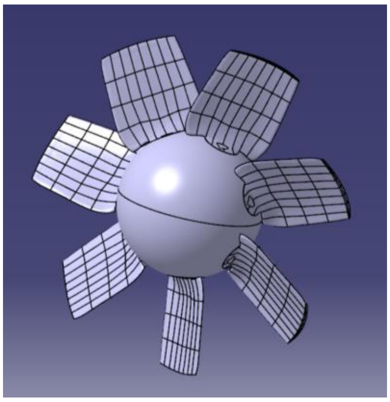

Figure 3 3D scan of the fan

The 3D-scanned impeller was not ideal for direct mesh generation, as it contained a large number of surfaces, which negatively affected the quality of the computational grid. The excessive surface detail resulted in a highly complex and irregular mesh, potentially leading to numerical instabilities and increased computational costs. To address this issue, the model was refined using cutting planes, simplifying its geometry while preserving the critical aerodynamic features of the impeller. This modification improved the mesh quality and ensured better computational performance without compromising the accuracy of the simulation.

3. CFD CALCULATIONS

In the computational domain, the entire air volume surrounding the fan was included to capture the full aerodynamic behavior. The mesh was generated using ANSYS Mesh, resulting in an unstructured numerical grid. While unstructured meshes offer flexibility in handling complex geometries, they are generally of lower quality compared to structured grids. To improve accuracy, the Contact Match method was applied, ensuring perfect alignment of the mesh nodes in the regions upstream and downstream of the fan. This technique significantly enhanced numerical stability by preventing interpolation errors at the interfaces between different flow regions. The final geometry is illustrated in Figure 3.

The outlet domain was carefully structured to facilitate the node-matching operation while simultaneously ensuring high mesh quality in the downstream region. This was particularly important because the research focused on two key measurement planes within the outlet domain. However, the Contact Match function does not support different mesh types or inflation layers within certain sectors. As a result, achieving a high-quality grid in the outlet region required careful adjustments to the meshing strategy.

One notable limitation arising from this approach was the absence of an inflation layer around the fan blades. Inflation layers are typically used to accurately resolve boundary layer effects, especially in turbulent flows. To compensate for this, the cell size in the blade region was reduced, allowing for a finer resolution of near-wall flow structures. The total computational grid consisted of 9.984 million cells, with 4.109 million cells dedicated to the structured outlet domain.

To overcome the problems of boundary conditions at the inlet of the fan, a large dome is created that incorporates the profiled free bell-mouth inlet. The outer perimeter of the dome represents the area where the total pressure is equal to the atmospheric pressure and where the velocity is equal to zero. The boundary conditions used are presented in Table 1.

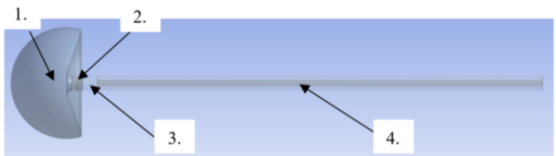

Figure 4 CAD model used for the simulation. 1 - inlet section, 2 - runner, 3 - non-structure mesh section outlet duct, 4 - structured mesh section outlet duct

Table 1. Boundary conditions

Surface	Boundary conditions	Input
Inlets	Total relative pressure	0 [Pa]
Outlets	Static relative pressure	0 [Pa]

For the simulation, the Shear Stress Transport (SST) turbulence model was chosen due to its favorable balance between computational cost and accuracy. The SST model effectively combines the k-ω model for near-wall regions and the k-ε model for free-stream turbulence, making it well-suited for complex aerodynamic flows such as those encountered in axial fan applications. While Large Eddy Simulation (LES) could potentially yield even more accurate results by resolving finer turbulence structures, its significantly higher computational cost makes it impractical for the current study. However, LES is being considered for future research to further improve the accuracy of the numerical predictions.

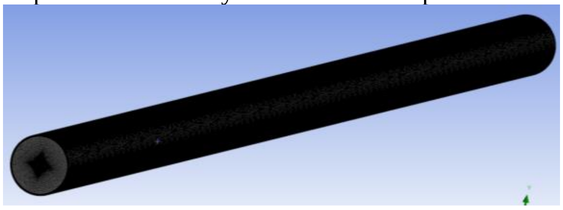

Figure 5 Mesh of the outlet pipe

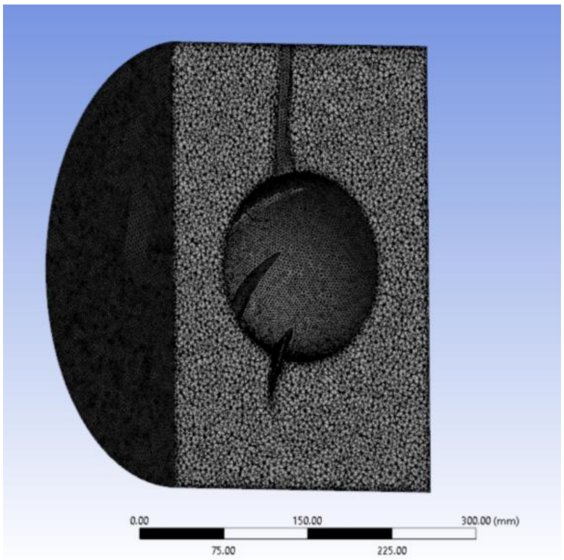

Figure 6 Mesh of the blade section

4. RESULTS AND DISCUSSION

The results obtained from the CFD simulation are compared with the experimental data acquired using a single-component Laser Doppler Anemometry (LDA) system which are obtained from Čantrak (2012)[1]. One of the primary advantages of CFD is its ability to provide detailed flow field analysis at any location within the simulation domain. However, for the sake of consistency and validation, the focus is placed on the same measurement planes that were analyzed during the experiments. These planes are located at *z/D* = 3.35 (referred to as Section 1) and *z/D* = 26.31 (referred to as Section 2) downstream of the fan outlet. These sections were chosen to capture both the near-field and far-field development of the swirling flow. [2]

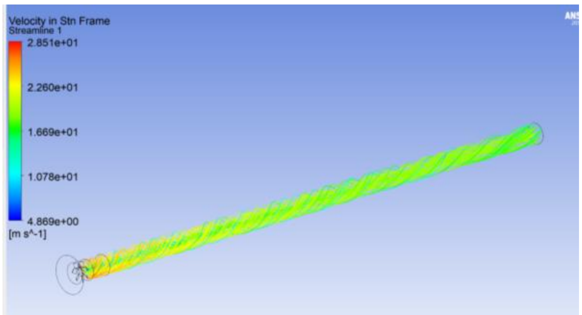

Figure 7 Representation of the streamlines starting from the outlet plane of the runner section until the end of the outlet

During the experimental measurements, all three velocity components were recorded in a polar-cylindrical coordinate system—the axial, radial, and circumferential velocity components. However, for this study, only the axial and circumferential velocity components are considered for comparison, as they are the most relevant in assessing the development of swirl flow and its impact on overall system performance. The circumferential velocity component plays a crucial role in influencing the axial velocity profile, as the presence of strong vortex structures alters the uniformity and distribution of axial flow. These vortex structures are clearly observed in the velocity contours obtained from the CFD simulation.

While the simulation successfully captures the general trend of the flow phenomena, discrepancies are observed in the magnitude of the velocity values when compared to experimental results. These differences can be attributed to multiple factors, including assumptions made in the numerical model, the idealized representation of the outlet duct, and the absence of certain geometric irregularities present in the experimental setup. Additionally, turbulence modeling limitations, such as the use of the SST model instead of LES, could contribute to minor deviations in predicted velocity distributions.

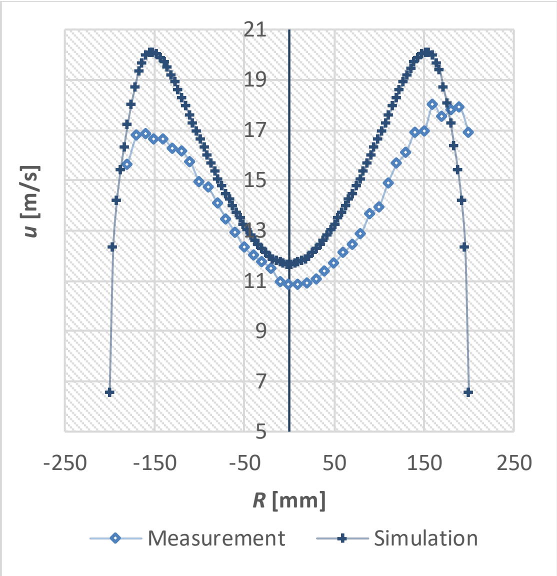

Figure 8 Axial velocity component in the section 1 plane [1]

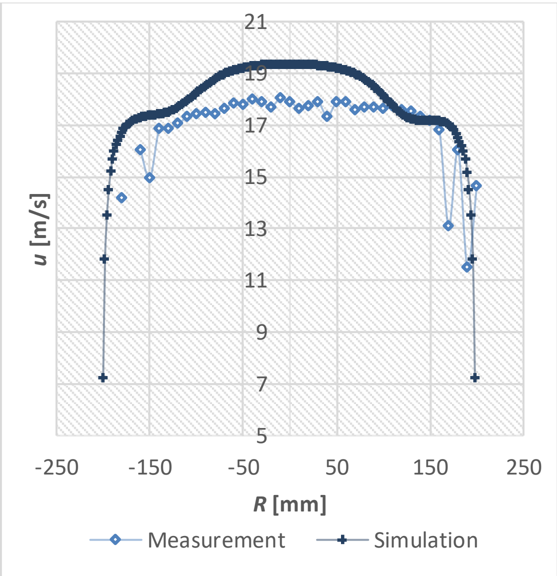

Figure 9 Axial velocity component in the section 2 plane [1]

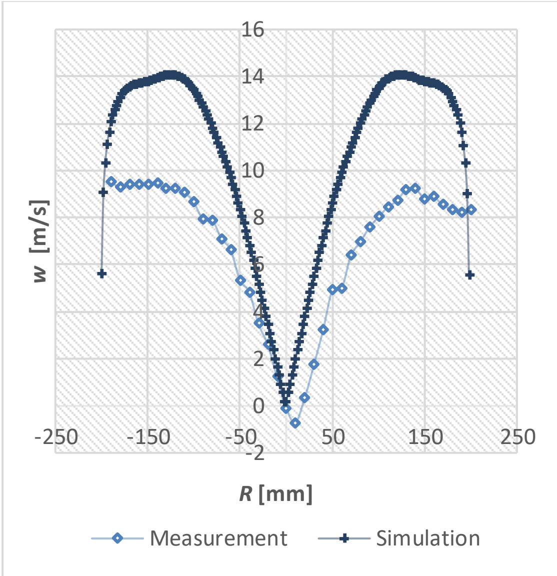

Figure 10 Circumferential velocity component in section 1 plane [1]

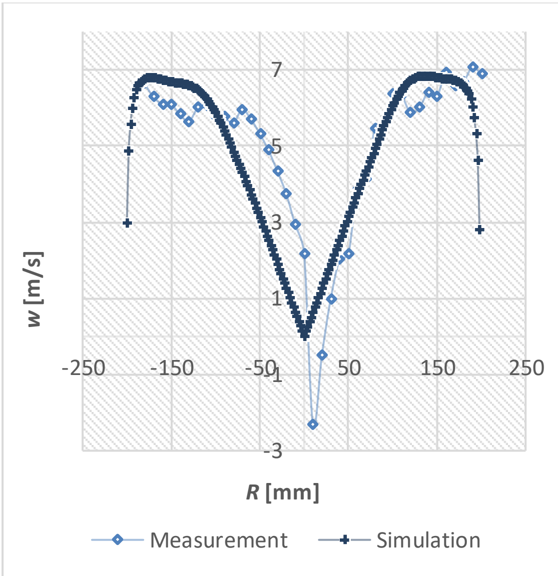

Figure 11 Circumferential velocity component in section 2 plane [1]

The simulation results indicate that the calculated velocity fields closely follow the trends observed in the experimental flow fields at both measurement sections. However, the agreement is more pronounced in Section 2 than in Section 1, suggesting that the numerical model better captures the flow characteristics further downstream.

One key observation is that the simulation treats the flow as nearly axisymmetric, whereas the experimental measurements reveal the opposite. In reality, the vortex core exhibited continuous shifts in position, indicating a more irregular, asymmetric behavior. This effect can be attributed to the presence of a significant radial velocity component, which the simulation does not fully capture. The shifting vortex core suggests the existence of complex secondary flow structures that are difficult to resolve using RANS-based turbulence models and may require more advanced modeling techniques, such as Large Eddy Simulation (LES) or Detached Eddy Simulation (DES), for a more accurate representation.

Apart from the symmetry discrepancy, differences in velocity magnitudes are also evident. The circumferential velocity component in Section 1 shows the highest deviations between the simulation and the measurements. This suggests that the near-field region of the swirling flow is not fully resolved in the numerical model, possibly due to mesh limitations or turbulence model assumptions. On the other hand, the smallest discrepancies in velocity magnitude are also observed in the circumferential component, but in Section 2, where only a slight radial shift is noted. This indicates that the swirl intensity diminishes further downstream, a trend that was also confirmed through experimental measurements.

Another notable difference between the simulation and the experimental results is the volumetric flow rate. The CFD simulation predicts a 10% higher flow rate compared to the measurements. This discrepancy leads to a general increase in axial velocity across all analyzed sections, as well as throughout the entire computational domain. The most likely cause of this overestimation is the assumption that the outlet duct is a perfect cylinder. In reality, the experimental test rig was constructed using commercially available components, which inherently contain manufacturing imperfections, creases, and slight misalignments. These irregularities introduce additional pressure losses and flow disturbances, ultimately reducing the measured flow rate in comparison to the idealized numerical model.

Figure 12 does not optimally illustrate the relationship between the experimentally measured velocity profiles and the simulated results. The most significant discrepancies between the two are observed near the vortex core, particularly in the circumferential velocity component. In this region, the simulation and experimental results diverge the most, indicating that the numerical model does not fully capture the local flow dynamics at the core of the swirling motion.

A consistent difference is also present in the axial velocity component, which suggests a discrepancy in the overall flow rate between the simulation and the experiment. As previously discussed, the CFD model predicts a higher volumetric flow rate, which directly influences the axial velocity distribution throughout the domain. This overestimation in axial velocity affects the overall agreement between the computational and experimental results.

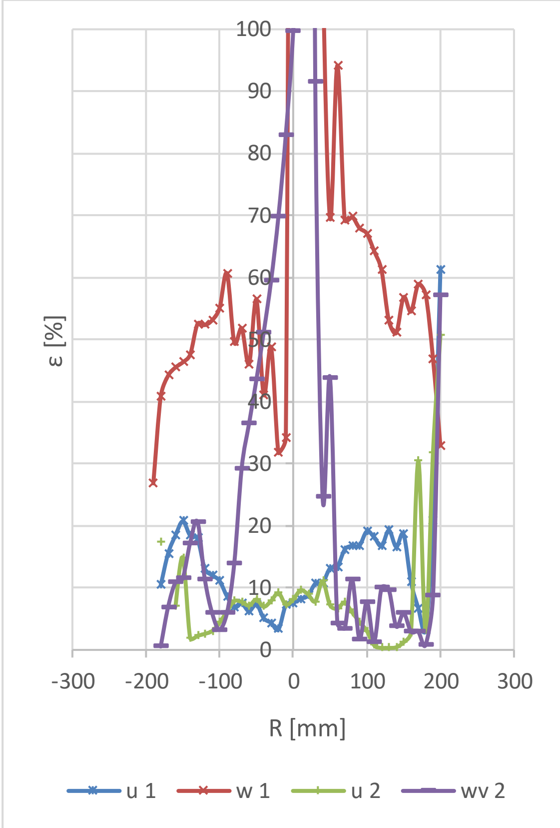

Figure 12 Relative difference between the simulated values and the experimental. Where u stands for axial, w for circumferential component of velocity, and 1 and 2 indicate the sections [1]

The relative difference in the circumferential velocity component near the vortex center can largely be attributed to the radial displacement of the vortex core. In the simulation, the vortex is treated as more stable and axisymmetric, while in the experiment, the vortex core exhibits continuous movement and shifting due to additional flow instabilities and secondary flow effects. If a correction method were applied to account for this radial displacement—such as realigning the vortex core locations between the simulation and the experiment—the resulting profiles would likely show better agreement.

5. SUMMARY

This paper presents a numerical comparison with a previously conducted experimental investigation on the flow characteristics in the ducted outlet of an axial fan without guide vanes. The study aims to evaluate the accuracy of computational fluid dynamics (CFD) simulations in replicating the complex flow structures observed in the experiment.

The results indicate a strong correlation in the general flow phenomena between the measured and simulated data, particularly in the solid-body circumferential velocity profiles. However, discrepancies in velocity magnitudes are evident, especially in the flow rate predictions. These differences are primarily attributed to the idealized representation of the flow field in the numerical model, which eliminates geometrical imperfections and surface irregularities present in the real experimental setup. The test rig components, including the duct and fan housing, contain minor manufacturing defects, creases, and misalignments, which introduce additional losses and distortions in the measured flow field—elements that are inherently absent in the simulation.

Given the complexity of both the flow dynamics and the geometric configuration, this study serves as an initial step toward a more comprehensive understanding of the intricate flow behavior downstream of the fan. Future research efforts could focus on expanding the computational domain, incorporating a more detailed representation of the fan, duct, and outlet section to better capture real-world effects.

Furthermore, enhancements in numerical modeling could lead to improved accuracy. Refining the computational mesh, particularly in regions of high velocity gradients, would help resolve finer flow structures. Additionally, exploring alternative turbulence models—such as Detached Eddy Simulation (DES) or Large Eddy Simulation (LES)—or modifying the existing turbulence approach could provide a more precise depiction of the unsteady vortex structures and turbulence intensities present in the flow.

By implementing these improvements, future studies can achieve higher accuracy and reliability in CFD predictions, ultimately contributing to a more detailed and practical understanding of swirl flow in axial fan systems.

ACKNOWLEDGEMENTS

This research was funded by the Ministry of Science, Technological Development and Innovation of the Republic of Serbia under the Agreement on financing the scientific research work of teaching staff at accredited higher education institutions in 2025, No. 451-03-137/2025-03/200105, and under the Agreement on financing Scientific research institutions in 2025, No. 451-03-136/2025-03/200105.

REFERENCES

[1] Čantrak Đ., 2012, “Analysis of the Vortex Core and Turbulence Structure Behind Fans in a Straight Pipe Using PIV, LDA and HWA Methods”, Ph.D. thesis, Univ. Belgrade, Fac. Mech. Eng., Belgrade.

[2] Čantrak Đ., Janković N., Ilić D., 2015, “Investigation of the Turbulent Swirl Flow in Pipe Generated by Axial Fans Using PIV and LDA Methods”, Theoretical and Applied Mechanics, Vol. 42, Iss. 3, pp. 211-222.

[3] Ilić D., Svorcan J., Čantrak Đ., Janković N., 2025, “Experimental and Numerical Research of Swirl Flow in Straight Conical Diffuser”, Processes, Vol .13, 182.

[4] Benišek M., 1979, “Investigation of the turbulent swirling flows in straight pipes, Ph.D. thesis, Univ. Belgrade, Fac. Mech. Eng., Belgrade.