

# MICROCYCLONES FOR POLLEN SEPARATION FROM AIR STREAM MIKROCIKLONI ZA IZDVAJANJE POLENA IZ VAZDUŠNE STRUJE

Andreja ŽIVKOV\*, Vladimir KOZOMORA, Nikola OLUŠKI, Slobodan TAŠIN, Maša BUKUROV, Iva GURANOV  
University of Novi Sad, Faculty of Technical Science, 21000 Novi Sad, Trg Dositeja Obradovića 6, Serbia

\*Correspondence: andreja.zivkov@uns.ac.rs

## ABSTRACT

The increasing concentration of airborne pollen presents significant challenges for public health and indoor air quality, resulting in development of efficient and compact separation technologies. This study focuses on the development and performance evaluation of microcyclones for the separation of pollen particles from air stream. Microcyclones utilize centrifugal forces generated by swirling airflow to separate particles based on their aerodynamic properties, offering advantages such as low maintenance, absence of filter media and continuous operation. The design methodology of the microcyclone is presented, including geometric parameters and air flow parameters selection to enhance pollen collection efficiency while minimizing pressure drop. In this study, a numerical simulation was performed to analyze the flow field prior to conducting the simulation of pollen particle separation from the air stream. The results indicated that properly designed microcyclones can achieve effective pollen removal with acceptable energy consumption, making them very suitable for different air pollutant analysis. The findings contribute to the advancement of microscale inertial separation devices for airborne allergen control. This study investigates an approach based on the scaling of standard cyclone geometries, combined with design optimization for fabrication using 3D printing technology.

**Keywords:** separation, microdevices, cyclones, microcyclone, pollen

## REZIME

Povećana koncentracija polena u vazduhu predstavlja značajan problem za javno zdravlje i kvalitet unutrašnjeg vazduha, što je dovelo do razvoja efikasnih i kompaktnih tehnologija za separaciju čestica. Ovaj rad se bavi razvojem i ispitivanjem mikrociklona za izdvajanje čestica polena iz vazdušne struje. Rad mikrociklona bazira se na centrifugalnoj sili, koja nastaje usled vrtložnog strujanja usled geometrije samog uređaja. Na osnovu ove sile moguće je obezbediti izdvajanje čestica polena usled njihovih aerodinamičkih karakteristika. Prednosti ovih uređaja su niska cena održavanja, nepostojanje filter medijuma i mogućnost kontinualnog rada. U radu je data geometrija samog uređaja, kao i odabrani radni parametri. U ovom radu izvršena je numerička simulacija radi analize strujnog polja pre sprovođenja simulacije separacije čestica polena iz vazdušne struje. Pravilno projektovani mikrocikloni mogu postići efikasno uklanjanje polena uz prihvatljivu potrošnju energije, što ih čini veoma pogodnim za analizu različitih zagađivača vazduha. Rezultati doprinose unapređenju mikrouređaja za inercijalnu separaciju namenjenih kontroli alergena u vazduhu. U radu je razmatran pristup zasnovan na skaliranju geometrije standardnih ciklona, uz optimizaciju konstrukcije za izradu primenom tehnologije 3D štampe.

**Cljučne reči:** separacija, mikrouređaji, cikloni, mikrocikloni, polen

## INTRODUCTION

Cyclones are inertial devices used for separating solid particles from an air stream in which they are dispersed. The separation principle is based on centrifugal force, which acts on the particles and enables their separation. The air stream enters the device tangentially, causing the formation of a vortex or cyclone flow. The vortex is rotating counterclockwise. It rotates around the cyclone axis inside the conical chamber. The air stream moves spirally downward along the cyclone walls, close to its outer radius. Due to centrifugal forces heavier particles can no longer follow the air stream and move toward the walls. Upon contact (collision) with the wall, the particles are separated from the air stream and slide downward under the effect of gravity. At the end of conical part of cyclone, flow direction of the clean air has been changed and moves upward along the cyclone axis and exits through the outlet pipe at the top. Figure 1 shows the movement of the air stream. The separation efficiency of a cyclone depends on particle size. If the particles have diameters larger than 10  $\mu\text{m}$ , the efficiency of these devices can reach up to 90%. However, these devices are not efficient for separating particles with small diameters. (Bukurov, 2009; Versteeg & Malalasekera, 1995)

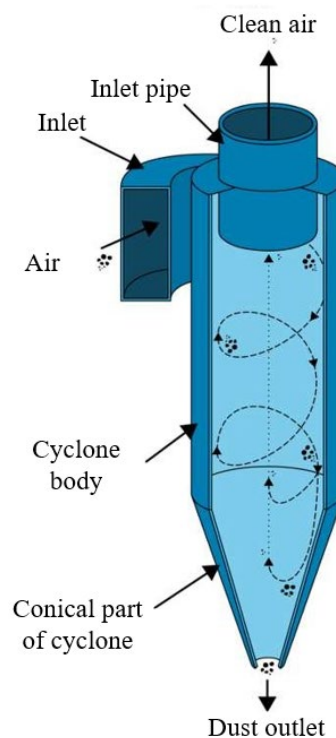


Fig. 1 Flow of the air in cyclone

**Nomenclature:**

- $Re$  (-) – Reynolds number
- $D_2$  (mm) – outer diameter
- $L_1$  (mm) – cylinder length
- $L_2$  (mm) – cone length
- $D_e$  (mm) – outlet diameter
- $H$  (mm) – inlet height
- $B$  (mm) – inlet width
- $D_d$  (mm) – dust outlet
- $L_3$  (mm) – dust outlet length
- $v$  (m/s) – velocity of the fluid
- $u_i$  (-) – flow variable
- $x_i$  (-) – coordinate
- $u_j$  (-) – flow variable
- $x_j$  (-) – coordinate
- $f_i$  (-) – external forces
- $p$  (Pa) – velocity of the fluid
- $t$  (s) – time
- $u_i'$  (-) – Reynolds stress
- $u_j'$  (-) – Reynolds stress
- $k$  (-) – turbulent kinetic energy
- $C_\mu$  (-) – model coefficient
- $C_{1\epsilon}$  (-) – model coefficient
- $C_{2\epsilon}$  (-) – model coefficient
- $\sigma_k$  (-) – model coefficient
- $\sigma_\epsilon$  (-) – model coefficient
- Greek symbols
- $\rho$  (kg/m<sup>3</sup>) – air density
- $\mu$  (Pas) – dynamic viscosity
- $\nu$  (m<sup>2</sup>/s) – kinematic viscosity
- $\nu_t$  (m<sup>2</sup>/s) – turbulent viscosity
- $\epsilon$  (-) – turbulent dissipation rate

**MODEL AND NUMERICAL SIMULATION**

The idea of developing a numerical simulation of a microcyclone originated from discussions with colleagues from BioSens, who attempted to construct such a device using pipette components for the separation of pollen particles from an air stream. However, this prototype was not a functional device and therefore could not be used for experimental validation. Nevertheless, it provided the initial motivation for the present study.

Accordingly, this work represents an initial step toward the development of a realistic and operational microcyclone by means of numerical modeling. The simulation was performed using the COMSOL Multiphysics software in collaboration with the BioSens team.

A numerical model was developed for a standard-type microcyclone. Microcyclones are scaled-down cyclone separators designed to enhance particle separation efficiency, particularly for fine particles. Their performance is strongly influenced by both inlet velocity and geometric configuration. Higher inlet velocities increase turbulence intensity, promoting particle migration toward the cyclone walls, while geometric parameters govern vortex formation and flow structure.

In this study, the geometry of the microcyclone was defined based on conventional design principles for standard cyclones, where all geometric parameters are determined as functions of the outer diameter  $D_2$  (Bukurov, 2009). The analyzed microcyclone is therefore assumed to be geometrically standard, with all dimensions scaled to an outer diameter of 10 mm. The corresponding geometric parameters are presented in Table 1.

Table 1. Standard microcyclone dimensions

Parameter	Value (mm)
Outer diameter $D_2$	10
Cylinder length $L_1$	$L_1=2 \cdot D_2=20$
Cone length $L_2$	$L_2=2 \cdot D_2=20$
Outlet diameter $D_e$	$D_e=1/2 \cdot D_2=5$
Inlet height $H$	$H=1/2 \cdot D_2=5$
Inlet width $B$	$B=1/4 \cdot D_2=2,5$
Dust outlet $D_d$	$D_d=1/4 \cdot D_2=2,5$
Dust outlet length $L_3$	$L_3=1/8 \cdot D_2=1,25$

Fig. 2 illustrates the geometry of a standard cyclone along (Bukurov,2009) with its corresponding 3D model.

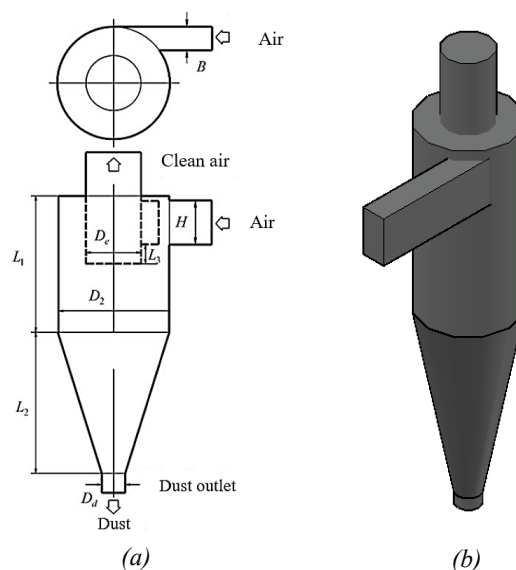


Fig. 2 Microcyclone geometry parameters: (a) in general; (b) model

Setting up a numerical simulation consists of several steps. The first step is drawing the geometry, i.e., defining the shape of the model within which fluid flow will be observed. The considered geometry needs to be divided into cells that form a mesh. The mesh should be as fine as possible to capture even the smallest changes in the observed fluid flow parameters. The mesh cells should not be excessively small to prevent unnecessary use of computational resources. After mesh generation, it is necessary to select the working fluid, i.e., the fluid that flows through the model and whose parameter variations are being analysed. After defining the model geometry and generating the mesh, it is necessary to select a solver. The solver represents one of the methods based on which, within the numerical simulation, the system of equations, namely the Navier–Stokes equations and the continuity equation, will be solved. Solvers in the COMSOL software are based on the finite volume method. Initial conditions must be specified for considered case. Once all elements of the numerical simulation have been defined, the solution process can begin. The software then proceeds to solve the corresponding equations, and after processing, concrete simulation results are obtained.

The selection of an appropriate turbulence model is a crucial step in the numerical simulation of cyclone and microcyclone flows, due to the presence of strong swirl, streamline curvature, and turbulence anisotropy. In engineering practice, Reynolds-averaged Navier–Stokes (RANS) models, particularly the  $k-\epsilon$

model, are widely used because of their robustness and relatively low computational cost. The  $k-\varepsilon$  model has been successfully applied in numerous studies of cyclone separators and provides satisfactory predictions of global flow characteristics. However, its underlying assumption of isotropic turbulence limits its accuracy in flows with strong rotation and anisotropy, which are typical for cyclone-type devices (Versteeg & Malalasekera, 1995; Ferziger & Perić, 1996).

An improvement within the RANS framework is offered by the  $SST k-\omega$  model, which combines the advantages of the  $k-\varepsilon$  and  $k-\omega$  formulations. This model provides enhanced accuracy in near-wall regions and better prediction of flow separation under adverse pressure gradients. For internal flows with significant wall influence, such as those in cyclone geometries, the  $SST k-\omega$  model is generally considered more reliable than the standard  $k-\varepsilon$  model, although it still cannot fully capture the anisotropic nature of strongly swirling turbulence.

For more detailed analysis of turbulent structures, Large Eddy Simulation (LES) represents a higher-fidelity approach, as it resolves large-scale turbulent motions directly while modeling only the smallest scales. This makes LES particularly suitable for capturing complex vortex structures, unsteady flow behavior, and secondary flow patterns in cyclone devices. The classical Smagorinsky subgrid-scale model is commonly used in LES; however, it is known to introduce excessive dissipation, especially in near-wall regions, which can reduce accuracy in wall-bounded flows (Smagorinsky, 1963; Pope, 2000).

In finite element-based computational environments, such as COMSOL Multiphysics, variational multiscale approaches have been developed to improve the performance of LES-type simulations. The Residual-Based Variational Multiscale (RBVM) model provides numerical stabilization by separating resolved and unresolved turbulent scales, offering improved robustness and reduced artificial dissipation compared to classical subgrid-scale models (Hughes et al., 1998; Bazilevs et al., 2007). Furthermore, the wall-weighted extension of this model (RBVMWW) enhances the treatment of near-wall turbulence, making it particularly suitable for small-scale devices where wall effects dominate the flow field.

Based on the above considerations, the RANS  $k-\varepsilon$  model was selected in this study as a computationally efficient approach for the initial analysis of the flow field and the influence of operating parameters. Although more advanced models, such as  $SST k-\omega$ , RSM, or LES-based approaches (e.g., RBVM and RBVMWW), may provide improved accuracy for strongly swirling flows, the chosen model is considered appropriate for this stage of research, where the primary objective is to establish a numerical framework and identify general flow trends. Future work will include the application of higher-fidelity turbulence models and experimental validation once adequate microfabrication precision is achieved.

Within this paper, it is assumed that the working fluid is air, which is an ideal gas, and that the flow is continuous, incompressible and isothermal; therefore, the numerical simulation is based on solving the Reynolds-averaged Navier–Stokes (RANS) equations. The flow inside the microcyclone was assumed to be in a steady state. The  $k-\varepsilon$  turbulence model was used due to its suitability for fully turbulent flows and low wall separation, as well as its capability to model isotropic turbulent behaviour. The numerical simulation is carried out by solving the following governing equations:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (1)$$

$$\rho \left( \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} \right) = - \frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial x_j^2} - \frac{\partial}{\partial x_j} (\rho u_i' u_j') \quad (2)$$

$$\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = P_k - \varepsilon + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial k}{\partial x_j} \right] \quad (3)$$

$$\frac{\partial \varepsilon}{\partial t} + u_j \frac{\partial \varepsilon}{\partial x_j} = C_{1\varepsilon} \frac{\varepsilon}{k} P_k - C_{2\varepsilon} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] \quad (4)$$

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \quad (5)$$

The inlet velocity was set to 4 m/s and 10 m/s, corresponding to Reynolds numbers  $Re=2656$  and  $Re=6640$  ensuring turbulent flow. The air flow was assumed to enter along the negative x-axis, and the outlet was set to atmospheric pressure.

Before the simulation, a computational mesh of the microcyclone was generated. The mesh was created automatically by COMSOL Multiphysics based on the geometry and turbulent flow model. A fine mesh was chosen to capture small variations in flow parameters, as this is a small-scale device.

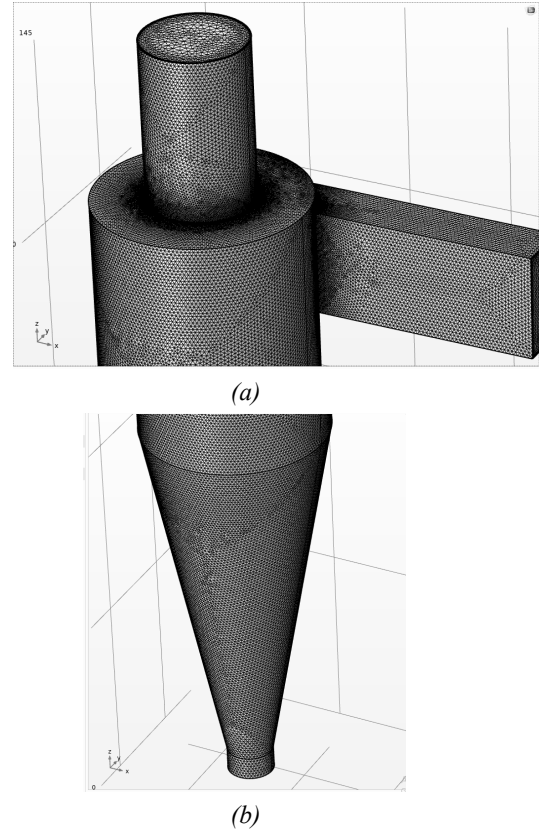


Fig. 3 Computational mesh: (a) inlet; (b) outlet

## RESULTS AND DISSCUSION

After completing the numerical simulation in the COMSOL software, the following velocity field distributions were obtained. In the paper the velocity field distributions are presented at an inlet velocity of 4 m/s and 10 m/s.

Based on the presented figures, it can be concluded that the fluid enters the microcyclone horizontally, along the x-axis in the direction opposite to the axis orientation. Subsequently, due to vortex formation, the flow turns and begins to move spirally downward along the outer surface of the microcyclone, first passing through the cylindrical section and then through the conical section. Upon reaching the conical part of the microcyclone, the fluid decelerates due to the weakening influence of the vortex. Following the streamlines, after the air reaches the bottom of the conical section, it turns upward and flows vertically along the axis of the microcyclone, all the way to the outlet surface.

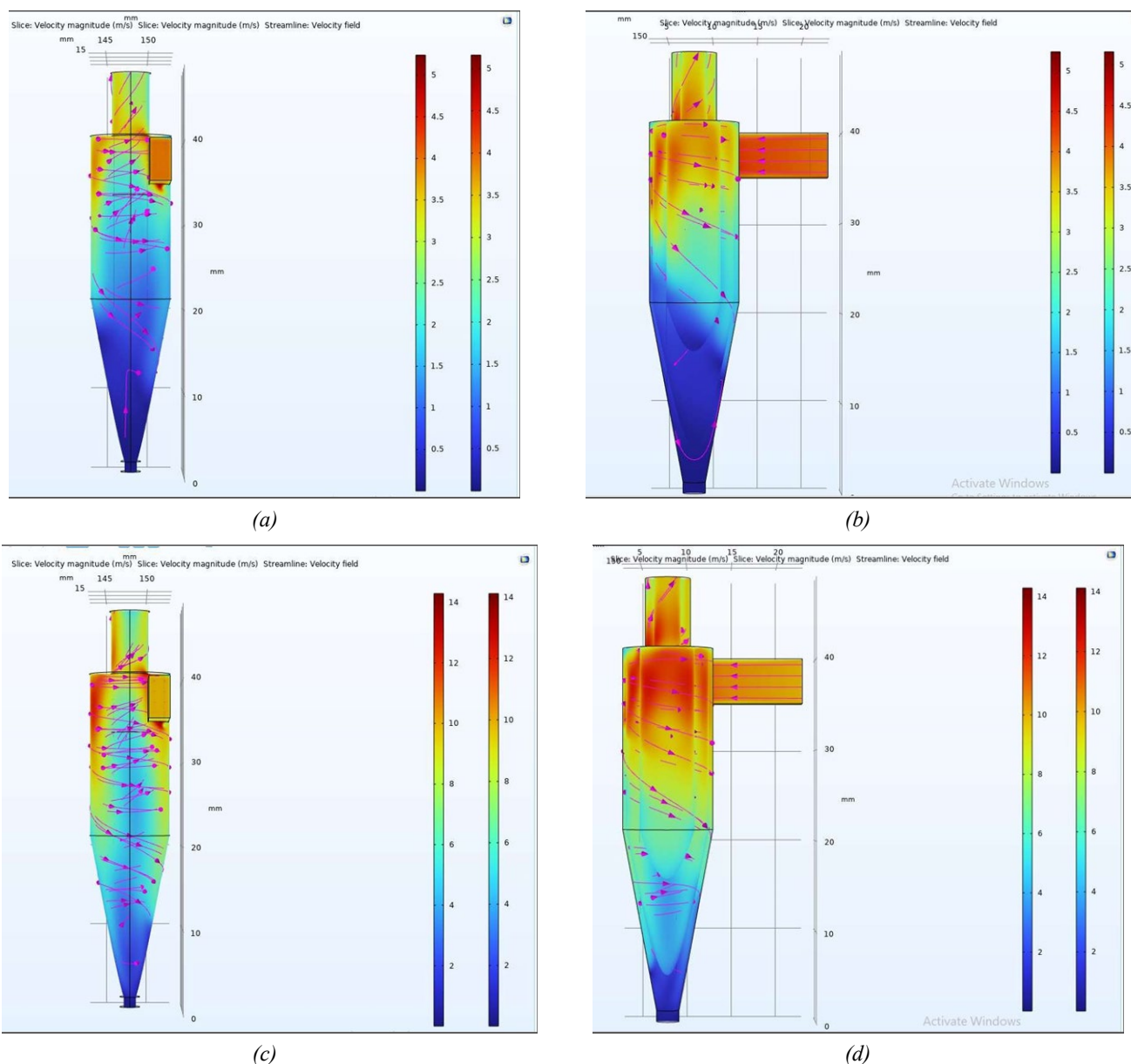


Fig. 4 Velocity field distribution: (a) *yz* plane inlet velocity 4 m/s; (b) *xz* plane inlet velocity 4 m/s; (c) *yz* plane inlet velocity 10 m/s; (d) *xz* plane inlet velocity 10 m/s

Based on the previously presented figures, figures 4a, 4b, 4c and 4d, it can be concluded that at higher inlet velocities, the velocities achieved inside the microcyclone are also higher. Applying an inlet velocity of 10 m/s results in stronger turbulence as well as more pronounced spiral downward motion of the fluid due to vortex formation. From the velocity distributions, it can be concluded that at higher inlet velocities, a larger amount of fluid passes through the microcyclone.

Specifically, the flow field has been quantified by analysing velocity distributions along characteristic cross-sections of the microcyclone for both investigated inlet velocities (4 m/s and 10 m/s), corresponding to Reynolds numbers  $Re = 2656$  and  $Re = 6640$ . The results indicate a clear increase in the magnitude of both axial and tangential velocity components with increasing inlet velocity.

For the case of 10 m/s, a significantly more pronounced vortex structure is observed, characterized by higher velocity gradients near the wall region and a stronger downward spiral motion compared to the 4 m/s case. This directly confirms the increased intensity of centrifugal effects, which are essential for particle

separation. In addition, the upward axial flow in the core region is more clearly developed at higher inlet velocity, indicating a more stable vortex core.

The comparison between the two operating regimes shows that increasing the inlet velocity leads to a higher overall flow rate through the device and a more intensive internal flow field, which is consistent with the expected behavior of cyclone-type separators.

The fabrication of a standard microcyclone at this scale requires high geometric fidelity, particularly in critical regions such as the inlet, vortex finder, and conical section, where small deviations can significantly affect flow structure and separation performance. At present, our available fabrication method is stereolithography (SLA), which, despite its advantages in rapid prototyping, exhibits known limitations at the microscale. These include insufficient resolution for small internal features, dimensional inaccuracies due to resin overcuring, and increased surface roughness, all of which can alter vortex formation and flow behavior within the device.

For this reason, experimental validation based on currently achievable geometries would introduce additional uncertainties related to manufacturing imperfections, making it difficult to isolate and assess the accuracy of the numerical model itself. Instead, this study focuses on establishing a reliable numerical framework for analyzing flow characteristics and guiding geometric optimization.

It should be noted that Reynolds-averaged Navier–Stokes (RANS) based CFD approaches, including the  $k-\epsilon$  turbulence model applied in this work, have been extensively used and validated in the analysis of cyclone separators. Previous studies have demonstrated that such models are capable of accurately capturing the main flow structures, including vortex formation, velocity distribution, and the influence of operating parameters, which supports their use in preliminary design and optimization stages.

## CONCLUSION

The aim of this paper was to analyse turbulent flow in a microcyclone in order to optimize its geometry and operating conditions. The main contribution of this work lies in the design of a micro-scale cyclone based on standard geometric proportions, capable of generating a flow field suitable for the separation of pollen particles from an air stream.

Based on the numerical results and available literature, it can be concluded that increasing the inlet velocity enhances vortex intensity and turbulence levels, thereby improving particle separation. Under such conditions, particles are less able to follow the gas streamlines, resulting in their migration toward the cyclone walls, followed by downward motion under the action of gravity. At the same time, higher inlet velocities enable increased air throughput, contributing to improved overall device performance.

The present study focuses on establishing a reliable characterization of the flow field and identifying operating conditions that ensure stable and sufficiently intense vortex formation, which represents a necessary first step prior to particle-phase modelling, given that cyclone performance is primarily governed by the structure of the flow field. Building upon this foundation, future research will address further geometric optimization, including refinement of inlet dimensions, overall scaling of the device, and the influence of inlet and outlet pressure conditions, with the aim of improving separation efficiency and operational stability. In addition, particle-tracking simulations and experimental validation will be conducted on a fabricated microcyclone once adequate manufacturing precision is achieved.

In this context, the present work represents an initial phase in the development of a microcyclone device, where numerical analysis precedes fabrication and serves as a basis for subsequent experimental investigations. Ongoing research is therefore directed toward improving SLA printing parameters and post-processing techniques in order to achieve the geometric precision required for manufacturing a standard microcyclone, while experimental validation will be carried out in subsequent studies once sufficient fabrication accuracy is attained, thereby completing the transition from numerical design to practical implementation.

**ACKNOWLEDGMENT:** This research has been supported by the Ministry of Science, Technological Development and Innovation (Contract No. 451-03-137/2025-03/200156) and the Faculty of Technical Sciences, University of Novi Sad through project “Scientific and Artistic Research Work of Researchers in Teaching and Associate Positions at the Faculty of Technical Sciences, University of Novi Sad 2025” (No. 01-50/295). The authors acknowledge the BioSens Institute for providing access to the COMSOL software.

## REFERENCES

- Bukurov, M., (2009). *Uredaji za mehaničko prečišćavanje vazduha*, FTN Izdavaštvo, Novi Sad
- Versteeg, H. u Malalasekera, W., (1995) *An introduction to Computational Fluid Dynamics – The Finite Volume Method*, Longman Group Ltd, Harlow, (accessed on 10.07.2025.), <https://gidropraktikum.narod.ru/Versteeg-Malalasekera.pdf>
- Ferziger, J. u Perić, M., (1996) *Computational Methodes for Fluid Dynamics*, Springer- Verlag Berlin Heidelberg, Germany
- Anderson, J., (1976) *Computational fluid dynamics-The Basics with Application*, McGraw-Hill Inc., New York, (accessed on 19.07.2025.), <https://elmoukrie.com/wp-content/uploads/2022/05/joel-h.-ferziger-milovan-peric-robert-l.-street-computational-methods-for-fluid-dynamics-springer-international-publishing-2020.pdf>
- Pandey, S., Saha, I., Prakash, O., Mukherjee, T., Iqbal, J., Roy, A. K., ... & Brar, L. S. (2022). CFD investigations of cyclone separators with different cone heights and shapes. *Applied Sciences*, 12(10), 4904., (accessed on 20.07.2025.), <https://www.mdpi.com/2076-3417/12/10/4904>
- Tran, T. T., & Nguyen, D. K. (2023). Numerical Investigation on the Performance of Cyclone Separators. *Journal of Technical Education Science*, 18(5), 25-34., (accessed on 20.07.2025.), [https://www.researchgate.net/publication/375065498\\_Numerical\\_Investigation\\_on\\_the\\_Performance\\_of\\_Cyclone\\_Separators](https://www.researchgate.net/publication/375065498_Numerical_Investigation_on_the_Performance_of_Cyclone_Separators)
- Qian, P., Ma, L., Liu, Y., & Zhang, Y. (2013, December). Numerical study of gas-liquid micro-cyclone separator flow field. In *AASRI Winter International Conference on Engineering and Technology (AASRI-WIET 2013)* (pp. 119-121). Atlantis Press., (accessed on 21.07.2025.), <https://www.atlantispress.com/proceedings/aasri-wiet-13/10896>
- Kang, Y. R., & Kwak, J. B. (2023). A numerical approach to characterize the efficiency of cyclone separator. *Machines*, 11(4), 440., (accessed on 21.07.2025.), <https://www.mdpi.com/2075-1702/11/4/440>
- Li, H. X., Gao, B. G., & Li, B. (2015). Numerical analysis of flow dynamics of cyclone separator used for circulating fluidized bed boiler. *Chemical Engineering Transactions*, 46, 991-996., <https://www.aidic.it/cet/15/46/166.pdf>
- [10] Kozić, Đ., Vasiljević, B., Bekavac, V., (1997). *Priručnik za termodinamiku*, Mašinski fakultet u Beogradu
- Vermande Paganel, T., Fabrice Alban, E., Cyrille, M. A., & Ngayihi Abbe, C. V. (2024). CFD Simulation of an Industrial Dust Cyclone Separator: A Comparison with Empirical Models: The Influence of the Inlet Velocity and the Particle Size on Performance Factors in Situation of High Concentration of Particles. *Journal of Engineering*, 2024(1), 5590437., (accessed on 22.07.2025.), [https://www.researchgate.net/publication/378978099\\_CFD\\_Simulation\\_of\\_an\\_Industrial\\_Dust\\_Cyclone\\_Separator\\_A\\_Comparison\\_with\\_Empirical\\_Models\\_The\\_Influence\\_of\\_the\\_Inlet\\_Velocity\\_and\\_the\\_Particle\\_Size\\_on\\_Performance\\_Factors\\_in\\_Situation\\_of\\_High\\_Concentr](https://www.researchgate.net/publication/378978099_CFD_Simulation_of_an_Industrial_Dust_Cyclone_Separator_A_Comparison_with_Empirical_Models_The_Influence_of_the_Inlet_Velocity_and_the_Particle_Size_on_Performance_Factors_in_Situation_of_High_Concentr)
- Chlebnikovas, A., Kilikevičius, A., Selech, J., Matijošius, J., Kilikevičienė, K., Vainorius, D., ... & Marcinkiewicz, J. (2021). The numerical modeling of gas movement in a single inlet new generation multi-channel cyclone separator. *Energies*, 14(23), 8092., (accessed on 22.07.2025.), <https://www.mdpi.com/1996-1073/14/23/8092>
- Utikar, R., Darmawan, N., Tade, M., Li, Q., Evans, G., Glenney, M., & Pareek, V. (2010). Hydrodynamic simulation of cyclone

- separators. *Computational fluid dynamics*, 11, 247., (accessed on 30.07.2025.), <https://www.scribd.com/document/77110046/Computational-Fluid-Dynamics>
- Blazek, J., (2005). *Computational Fluid Dynamics: Principles and Applications (Second Edition)*, Elsevier Science, (accessed on 30.07.2025.), [https://dl.amobbs.com/bbs\\_upload782111/files\\_46/ourdev\\_680516HBAK3D.pdf](https://dl.amobbs.com/bbs_upload782111/files_46/ourdev_680516HBAK3D.pdf)
- Menter, F. R., (1994) Two-equation eddy-viscosity turbulence models for engineering applications, *AIAA Journal*, 32(8), pp. 1598–1605, (accessed on 27.03.2026.), [https://www.academia.edu/34703063/Two\\_Equation\\_Eddy\\_Viscosity\\_Turbulence\\_Models\\_for\\_Engineering\\_Applications](https://www.academia.edu/34703063/Two_Equation_Eddy_Viscosity_Turbulence_Models_for_Engineering_Applications)
- Wilcox, D. C., (2006) *Turbulence Modeling for CFD*, DCW Industries, La Canada, California, (accessed on 27.03.2026.), [Turbulence Modeling for CFD PDF](#)
- Smagorinsky, J., (1963) General circulation experiments with the primitive equations: I. The basic experiment, *Monthly Weather Review*, 91(3), pp. 99–164, (accessed on 27.03.2026.), [https://journals.ametsoc.org/view/journals/mwre/91/3/1520-0493\\_1963\\_091\\_0099\\_gcewtp\\_2\\_3\\_co\\_2.xml](https://journals.ametsoc.org/view/journals/mwre/91/3/1520-0493_1963_091_0099_gcewtp_2_3_co_2.xml)
- Pope, S. B., (2000) *Turbulent Flows*, Cambridge University Press, Cambridge, (accessed on 27.03.2026.), [Turbulent Flows PDF](#)
- Hughes, T. J. R., Feijóo, G. R., Mazzei, L., & Quincy, J. B., (1998) The variational multiscale method—a paradigm for computational mechanics, *Computer Methods in Applied Mechanics and Engineering*, 166(1–2), pp. 3–24, (accessed on 27.03.2026.), <https://www.sciencedirect.com/science/article/pii/S0045782598000796>
- [20] Bazilevs, Y., Calo, V. M., Cottrell, J. A., Hughes, T. J. R., Reali, A., & Scovazzi, G., (2007) Variational multiscale residual-based turbulence modeling for large eddy simulation of incompressible flows, *Computer Methods in Applied Mechanics and Engineering*, (accessed on 27.03.2026.), <https://www.sciencedirect.com/science/article/pii/S0045782507003027>

Received: 27. 01. 2026.

Accepted: 01. 04. 2026