Slovak Society of Chemical Engineering
Institute of Chemical and Environmental Engineering
Slovak University of Technology in Bratislava

PROCEEDINGS

51st International Conference of the Slovak Society of Chemical Engineering SSCHE 2025

Hotel DRUŽBA

Jasná, Demänovská Dolina, Slovakia

May 27 - 30, 2025

Editors: Assoc. Prof. Mário Mihaľ

ISBN: 978-80-8208-158-2, EAN: 9788082081582

Published by the Faculty of Chemical and Food Technology Slovak Technical University in Bratislava in Slovak Chemistry Library for the Institute of Chemical and Environmental Engineering; Radlinského 9, 812 37 Bratislava, 2024

Kaczmarzyk, P., Małozięć, D., Ziegler, B.: Numerical method of testing the flow parameters of positive pressure ventilators on the ISO 5801 test stand, Editors: Mihaľ, M., In *51st International Conference of the Slovak Society of Chemical Engineering SSCHE 2025*, Jasná, Demänovská Dolina, Slovakia, 2025.

Numerical method of testing the flow parameters of positive pressure ventilators on the ISO 5801 test stand

Piotr Kaczmarzyk [1], Daniel Małozięć [1], Bartosz Ziegler [2]

[1] *Scientific and Research Centre for Fire Protection, National Research Institute, 05-420 Józefów, Poland*
[2] *Institute of Thermal Energy, Faculty of Environmental and Energy Engineering, Poznań University of Technology, 60-965 Poznań, Poland*

e-mail: pkaczmarzyk@cnbop.pl

Key words: positive pressure ventilators, airflow, iso 5801, CFD, LES

Positive pressure ventilators are an important tool used in rescue operations by fire protection units. The aim of this article is to assess the potential of using LES-type analyses for CFD simulations (in the Fire Dynamics Simulator software) to evaluate the flow parameters of a positive pressure ventilator under the conditions of the PN-EN ISO 5801 method. The article presents a comparative analysis that enables an assessment of the level of convergence between the volumetric airflow rate and pressure obtained in full-scale experimental studies and numerical CFD analysis. For the airflow rate [m³/h], a convergence of 1.4% was achieved, while for pressure [Pa], the convergence was 17%. The analysis demonstrated that the LES model is an appropriate tool for reflecting the conditions specified by the PN-EN ISO 5801 method. The CFD simulation method described in the publication may serve as a useful tool for manufacturers of positive pressure ventilators, enabling the implementation of technological pre-tests without the need for costly laboratory experiments.

Numerical method of testing the flow parameters of positive pressure ventilators on the ISO 5801 test stand

Piotr Kaczmarzyk[1], Daniel Małozięć[1], Bartosz Ziegler[2]

[1] *Scientific and Research Centre for Fire Protection, National Research Institute, 05-420 Józefów, Poland*
[2] *Institute of Thermal Energy, Faculty of Environmental and Energy Engineering, Poznań University of Technology, 60-965 Poznań, Poland*

e-mail: pkaczmarzyk@cnbop.pl

Key words: positive pressure ventilators, airflow, iso 5801, CFD, LES

Abstract

Positive pressure ventilators are an important tool used in rescue operations by fire protection units. The aim of this article is to assess the potential of using LES-type analyses for CFD simulations (in the Fire Dynamics Simulator software) to evaluate the flow parameters of a positive pressure ventilator under the conditions of the PN-EN ISO 5801 method. The article presents a comparative analysis that enables an assessment of the level of convergence between the volumetric airflow rate and pressure obtained in full-scale experimental studies and numerical CFD analysis. For the airflow rate [m³/h], a convergence of 1.4% was achieved, while for pressure [Pa], the convergence was 17%. The analysis demonstrated that the LES model is an appropriate tool for reflecting the conditions specified by the PN-EN ISO 5801 method. The CFD simulation method described in the publication may serve as a useful tool for manufacturers of positive pressure ventilators, enabling the implementation of technological pre-tests without the need for costly laboratory experiments.

Introduction

Positive pressure ventilators are flow machines used to force airflow through buildings, escape routes and other volumes when required for rescue operations. They create a jet of atmospheric air, which momentum can be used to create desired mass flow or overpressure. For a Newtonian fluid (like air), with negligible compressibility and no heat transfer, a set of mass and momentum transport equations known as continuity (equation 1) and Navier-Stokes (equation 2) equations [X1] describe the flow physics sufficiently.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0 \tag{1}$$

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \mu \nabla^2 \mathbf{u} + \rho \mathbf{g} \tag{2}$$

The assumption of negligible density changes is related directly to flow velocity as it can be shown that relative density changes are related to the Mach number (dimensionless number describing local ratio of flow velocity to local velocity of sound). For a steady isentropic flow along a streamline:

$$\frac{\partial \rho}{\rho} = \frac{\partial \rho}{\partial p} \cdot \frac{\partial p}{\rho} = \frac{1}{a^2} u \, du = \frac{u^2}{a^2} \frac{\partial u}{u} = M^2 \frac{\partial u}{u} \tag{3}$$

In equation 3: a is the speed of sound and M – is the Mach number. As the relative change in density is related to relative change in flow velocity by a factor of M^2, for small Mach numbers (by convention M<0.3) the density changes can be assumed negligible [1].

As the jet region of a positive pressure ventilator in general doesn't get near the M = 0.3 limit, numerical methods for obtaining approximate solutions for equations 1-2, with some additional attention to turbulent effects can be used to estimate the effects of positive pressure ventilator induced flow.

With regard to the use of positive pressure ventilators, it should be noted that they are an important tool employed in rescue operations. These devices are used, among other things, for smoke removal from buildings using various ventilation techniques such as PPV (positive pressure ventilation) or NPV (negative pressure ventilation) [2-3]. These techniques make it possible to remove toxic thermal decomposition products from fire-affected areas [4-5].

Kaczmarzyk et al. (2023) analysed the impact of the positioning distance of a positive pressure ventilator on flow parameters such as volumetric airflow rate and pressure in a real building environment [6]. The experimental results showed significant differences in performance depending on the type of unit. The conventional ventilator achieved the highest volumetric flow rate when positioned at a distance of 1 meter, whereas the turbo model, equipped with a flow straightener mounted on the impeller, was most effective at a distance of 5 meters.

Panindre et al. in 2017 conducted a study analysing the impact of inlet size on the efficiency of positive pressure ventilation (PPV) – a technique based on ventilating a building through an inlet using a portable fan placed in front of it [7]. The research showed that airflow intensity can be increased by installing a mobile smoke curtain at the upper part of the door frame. Similarly, Lambert et al. in 2014 [8] carried out a series of experimental studies aimed at determining the optimal positioning of PPV fans within a stairwell. They observed that when using a single fan, its efficiency increases as the distance from the entrance door decreases. The authors also noted that increasing the number of fans leads to greater airflow, and the most effective arrangement of two fans is achieved when they are positioned in a V-shape with an internal angle of 60°. The literature also includes articles describing simulations of airflow parameters of portable fans under real operating conditions. Kerber (2006) used the FDS program to analyse the air stream velocity profile in open flow, examining the impact of various positioning distances of a positive pressure ventilator. In his study, he achieved agreement between experimental results and CFD simulations at the following levels: 8.7% for a distance of 1.8 m, 14.7% for 2.4 m, and 3.1% for 3.1 m [9]. A similar comparison was conducted by Weinschenk et al. in 2011 [10]. In their publication, they assessed the usefulness of both simple analytical methods and CFD simulations for predicting airflow parameters generated by a positive pressure ventilator in a building. For the analysed structure, the level of agreement between CFD simulations and actual measurements was 2.5% for pressure and 9.9% for mass flow rate. Additionally, Wang et al. in 2024 conducted experimental and numerical analyses for an axial fan combined with an LFTHE (louvered fin-and-tube heat exchanger) [11]. Regarding flow parameters, the researchers achieved a convergence level between experiments and CFD simulations of 8.9% for flow rate and 8.3% for pressure. However, the literature lacks studies describing a comparative analysis between tests conducted using standardized test setups and CFD analyses aimed at reproducing the conditions of the method.

The aim of this article is to evaluate the potential of using LES-type analyses for CFD simulations (in the Fire Dynamics Simulator software) to assess the flow parameters of a positive pressure ventilator under the conditions of the PN-EN ISO 5801 standard [12]. The article presents a comparative analysis that allows for the assessment of the level of agreement between the volumetric airflow rate and pressure obtained from full-scale experimental tests (conducted on a PN-EN ISO 5801 test stand) and numerical CFD analysis (performed using the Fire Dynamics Simulator).

Material and methods

Experimental testing method in compliance with PN-EN ISO 5801 requirements

The study used a turbo-type firefighting ventilator, model GX 350. The tested ventilation unit is a commonly used tool by fire protection services in Poland and across Europe. The positive pressure ventilator was powered by a gasoline engine with a power output of 4.1 kW and a displacement of 196 cm^3.

The flow parameter characteristics (volumetric airflow rate and pressure) were tested on a test stand (fig. 1) built in accordance with the requirements of the PN-EN ISO 5801 standard [12]. According to PN-EN ISO 5801, the constructed test stand is classified as type B, i.e. free inlet, ducted outlet. The diameter of the flow duct was 475 mm. The volumetric airflow rate (for 10 damper position configurations) was determined based on dynamic pressure measurements using a Prandtl tube. Traversing of the dynamic pressure profile (14 points located across the cross-sectional area of the duct) was carried out using the equal area method in accordance with ISO 3966 [13]. Measurements of static pressure were performed using pressure transducers (Setra, with a measuring range of ±1 inch of water column), which were mounted directly on the connection nozzle (conical inlet) of the test stand (fig. 1).

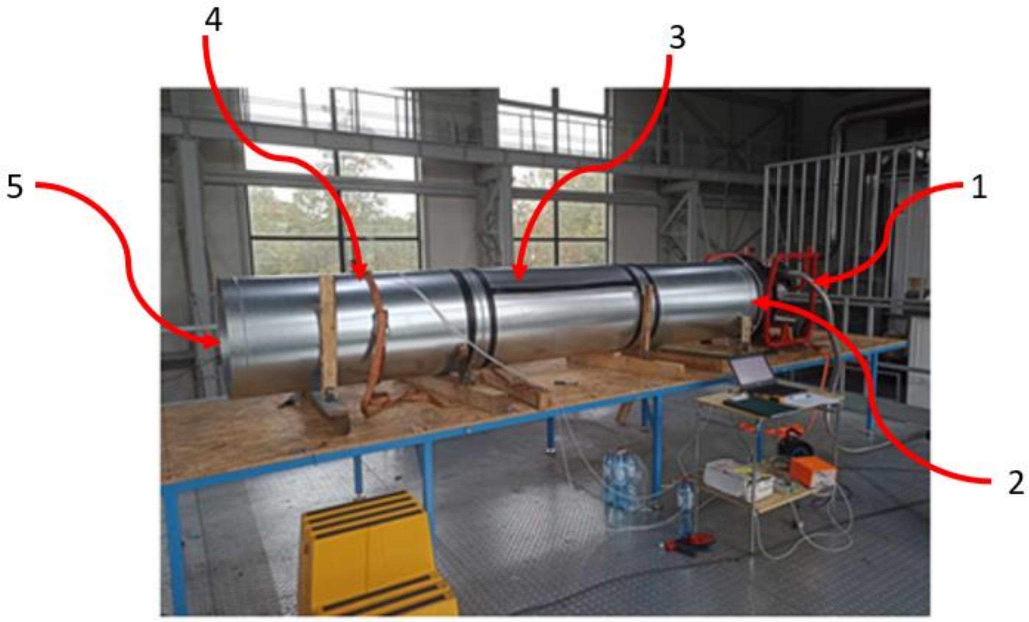

Fig. 1. Test stand for flow parameter measurements according to PN-EN ISO 5801, where: 1 – positive pressure ventilator, 2 – connection nozzle and static pressure measurement point (annular system), 3 – star-type flow straightener, 4 – velocity profile traversing point inside the duct using a Prandtl tube, 5 – throttle device.

The acquisition frequency for flow velocity and pressure parameters was 10 Hz, and the measurement duration was 30 seconds. During the tests, the positive pressure ventilator operated at maximum rotational speed. The experimental tests were conducted inside a test hall with a volume of 1500 m^3. Environmental conditions were monitored throughout the testing process and were as follows: temperature (15 ± 2°C), humidity (50 ± 3%), and pressure (1003 ± 10 hPa).

Method for assessing the air jet velocity profile of the positive pressure ventilator

To obtain reliable flow data for use in the CFD numerical simulation model with the Fire Dynamics Simulator software, measurements of the air jet velocity profile were conducted in the immediate vicinity of the rotor of the positive pressure ventilator. The velocity profile measurement in the direct area of the ventilator was carried out for the same unit that had previously been tested in accordance with the requirements of PN-EN ISO 5801.

The measurements were carried out using a modified equal-area method in accordance with ISO 3966 [13]. The modification to the standard procedure involved excluding the central hub area of the fan (the dead zone), within which traversing of dynamic pressure field was not performed. The velocity profile was measured using a Prandtl tube and Setra pressure transducers. Additional measuring tools, such as a tape gauge, calliper, and protractor, were also used during the tests. Measurements were taken in a symmetrical arrangement relative to the rotor axis (5 points on each side) [14]. The volumetric airflow rate generated by the positive pressure ventilator was assessed based on the product of the average flow velocity and the surface area of the fan rotor.

CFD analysis method

The numerical flow analysis (CFD) was carried out using Fire Dynamics Simulator software, version 6.9.1 [15]. This program is dedicated to the simulation of low-velocity turbulent flows and employs the Large Eddy Simulation (LES) method for turbulence modelling [16]. FDS solves the LES equations (temporally and spatially filtered Navier-Stokes equations), which allows for the direct modelling of larger turbulent structures that are critical for mass and momentum transport within the flow. In the conducted simulation, the primary driving force of the flow was the positive pressure ventilator. An isothermal flow model was used in the simulation. The Pyrosim software, version 2024.2.1209, was used to visualize the test conditions. The air jet generated by the ventilator was defined using the "Velocity Patch" function [17], which allows for the assignment of a non-uniform velocity profile on the surface of the fan rotor.

The volumetric airflow rate of the positive pressure ventilator was defined based on the velocity profile measurements of the air jet in the immediate vicinity of the fan rotor, as previously described (12,600 m^3/h). The airflow on the rotor surface was measured and controlled using the "volume flow in the gas phase" function [18]. For the purpose of the FDS simulation, aimed at replicating the test conditions in accordance with PN-EN ISO 5801 standards, a 3D model of the test setup, its surrounding environment, and the positive pressure ventilator was created. The spatial model was built on a Cartesian computational mesh divided into 1,267,200 cells, each with dimensions of 0.01 x 0.01 x 0.01 meters. The model was defined to reflect the actual working conditions of the test stand and the ventilator. All elements of the model, including the ventilator housing and the steel structure of the flow duct, were created with a precision corresponding to the grid cell resolution, i.e. 1 cm. Other model parameters included: atmospheric pressure (1013.3 hPa), ambient temperature (15°C), primary material used for model construction – steel, relative humidity (50%), and atmospheric temperature gradient (0.03 K/m). The model was equipped with flow parameter measurement points (Fig. 2), including volumetric airflow rate [m^3/h] in the outlet area of the flow duct (measured using the "volume flow in the gas phase" function). Static pressure [Pa] was measured using transducers located in the connection nozzle of the tested ventilator.

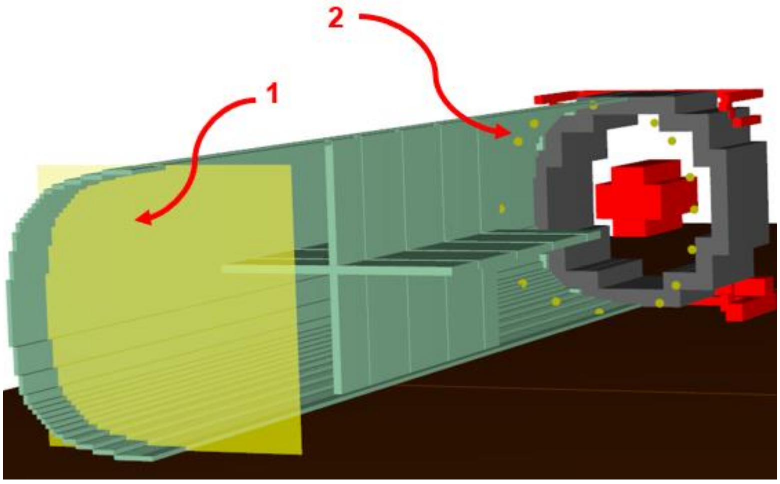

Fig. 2. Visualization of the numerical model developed for the simulation – PN-EN ISO 5801
(1 – measurement plane for volumetric flow rate "volume flow in the gas phase";
2 – distribution of static pressure measurement points)

As part of the conducted work, a grid convergence analysis was performed. Resolutions of 1 cm, 0.5 cm, and 0.25 cm were tested. The analysis showed that a grid resolution of 1 cm is optimal – it allows for the assessment of flow parameters with satisfactory accuracy without requiring excessive computational power. To verify the simulation, an evaluation of the numerical stability was also carried out.

Result and analysis

A comparative analysis of the flow parameters generated by the tested positive pressure ventilator under real-world conditions and within the CFD simulation is presented in Fig. 3. Meanwhile, a visualization of the airflow within the PN-EN ISO 5801 flow duct during the CFD simulation is shown in Fig. 4. The assessment of flow parameters under experimental conditions was carried out for 10 different throttle device opening configurations [6], whereas the numerical analysis was performed for a single throttle position (full-open configuration).

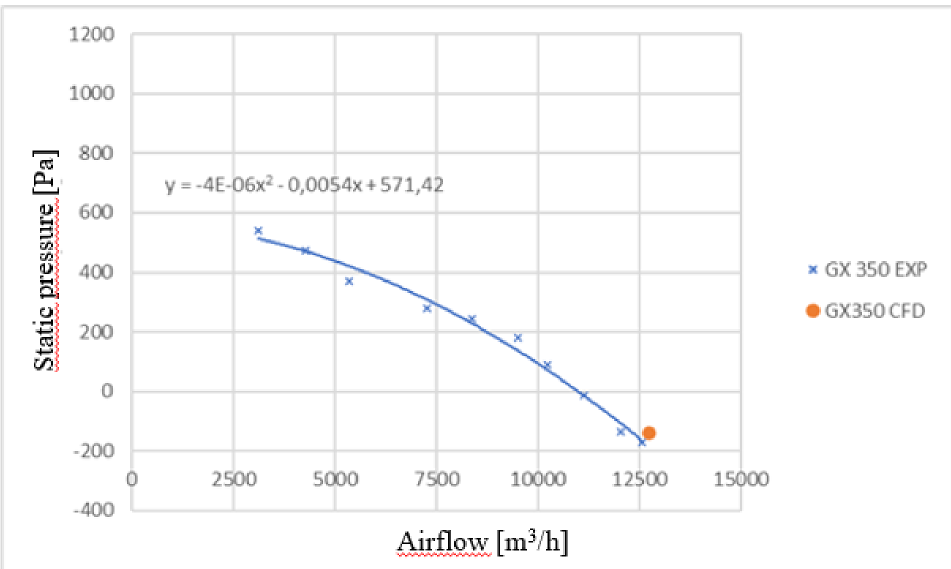

Fig. 3. Characteristics of volumetric airflow rate and pressure obtained from real-world testing and CFD simulation (test conditions in accordance with PN-EN ISO 5801 requirements)

Analysing the characteristic point corresponding to the fully open throttle position (the condition most closely resembling the real operating conditions of the positive pressure ventilator), the experimental tests showed that the ventilator generated a volumetric airflow rate of 12,569.4 m³/h and a pressure of -169.4 Pa. The volumetric flow rate produced during the CFD simulation was 12,744.0 m³/h, with a pressure of -140.5 Pa. Based on the analysis, the level of agreement between the experimental results and the CFD simulation, expressed as a percentage difference, was approximately 1.4% for airflow rate and 17% for pressure.

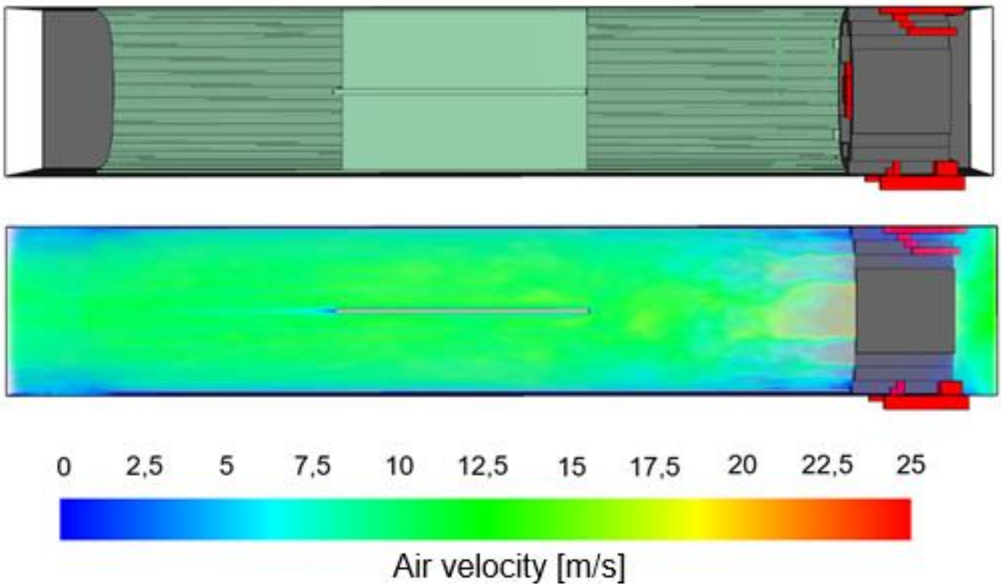

Fig. 4. CFD simulation visualization (velocity profile distribution inside the duct) based on flow parameter testing in accordance with PN-EN ISO 5801

The obtained values of flow parameters are consistent with the results of experimental studies and CFD simulations reported by other researchers. Fritsche et al., in 2018, conducted experimental tests and CFD analysis of the volumetric airflow generated by a fan in a closed duct [19]. During the tests, the fan moved an airflow of approximately 14,000 m³/h through the duct.

When describing the obtained research results, it should also be emphasized that the software allowed for an accurate representation of flow phenomena within the duct. This is evidenced by the negative pressure values recorded by the sensors in both the experiment and the CFD simulation, which were caused by the presence of local flow separation and turbulence in the near-wall regions of the duct. This conclusion supports the validity of using the FDS software to evaluate flow parameters with the applied methods. Regarding the 17% discrepancy in pressure values, it should be noted that it was caused by the non-uniformity of the airflow field, which resulted from surface irregularities within the duct, particularly in areas where the pressure transducers were installed. Moreover, the CFD simulation using the Fire Dynamics Simulator is somewhat simplified and limited to certain flow functionals, which makes it practically impossible to fully replicate local flow non-uniformities. In the context of using a positive pressure ventilator, it is also important to highlight the limitations of the PN-EN ISO 5801 method. This standard does not allow for the determination of the most critical parameters of the airflow generated by mobile ventilators, such as the velocity profile characteristics and its distribution changes (in free flow), as well as the related geometric positioning parameters (such as rotor tilt angle and distance from the doorway), which are crucial for identifying the most effective operating conditions of a mobile ventilation unit.

Conclusions:

The evaluation of airflow parameters generated by a mobile ventilator is a crucial factor in confirming its operational effectiveness. This assessment can be carried out using CFD tools based on LES-type analyses. In this study, a comparative analysis was conducted to assess the consistency of flow parameters (volume flow rate and pressure). The comparison included both full-scale experimental tests and CFD simulations performed on a test bench in accordance with the PN-EN ISO 5801 standard. Under the specified test conditions, the convergence between experimental and simulation results was as follows: volume flow rate – 1.4%, and pressure – 17%. The achieved level of convergence confirms the potential for using LES-type analyses by manufacturers of positive pressure ventilators to conduct preliminary technological testing without the need for costly laboratory experiments. It is also noteworthy that the CFD analysis enabled accurate replication of airflow within the test duct. This is evidenced by the negative pressure values recorded both during the experiment and the CFD simulation, resulting from local flow separation and turbulence near the duct walls. The authors express their intent to continue developing research on the applicability of LES-based analyses in relation to other methods used for evaluating airflow parameters of positive pressure ventilators, such as ANSI/AMCA 240-22.

Literature

1. White, F. M. (2015). Fluid Mechanics, University of Rhode Island,
2. Ziesler P. S., Gunnerson F. S., Williams S. K., Advances in positive pressure ventilation: Live fire tests and laboratory simulation, ”Fire Technology” 1994, 30(2), 269–277, https://doi.org/10.1007/BF01040006.
3. Garcia K., Kauffmann R., Schelble R., Positive pressure attack for ventilation & firefighting, PennWell Books 2006
4. Krawiec, P., Warguła, Ł., Małozięć, D., Kaczmarzyk, P., Dziechciarz, A., & Czarnecka-Komorowska, D. (2020). The Toxicological Testing and Thermal Decomposition of Drive and Transport Belts Made of Thermoplastic Multilayer Polymer Materials. *Polymers*, *12*(10), 2232. https://doi.org/10.3390/polym12102232
5. Krawiec, P., Warguła, Ł., Czarnecka-Komorowska, D. *et al.* Chemical compounds released by combustion of polymer composites flat belts. *Sci Rep* **11**, 8269 (2021). https://doi.org/10.1038/s41598-021-87634-9
6. Kaczmarzyk. P. et al. Influence of the Positive Pressure Ventilator Setting Distance in Front of the Doorway on the Effectiveness of Tactical Mechanical Ventilation in a Multistory Building. *Appl. Sci.* 13. 5536 (2023).
7. Panindre, P., Mousavi, N. S., & Kumar, S. (2017). Improvement of Positive Pressure Ventilation by optimizing stairwell door opening area. *Fire Safety Journal*, *92*, 195-198.
8. Lambert, K., & Merci, B. (2014). Experimental study on the use of positive pressure ventilation for fire service interventions in buildings with staircases. Fire Technology, 50(6), 1517-1534
9. Kerber. S. Evaluation of the Ability of Fire Dynamic Simulator to Simulate Positive Pressure Ventilation in the Laboratory and Practical Scenarios. *Natl. Inst. Stand. Technol. Gaithersburg MD USA* (2006).
10. Weinschenk. C.. Beal. C. M.. & Ezekoye. O. A. (2011). Modeling fan-driven flows for firefighting tactics using simple analytical models and CFD. *Journal of fire protection engineering*. 21(2). 85-114.
11. Wang, Z., Liu, J., Wang, H., Zhang, B., Liu, W., Özdemir, E. D., & Aksel, M. H. (2024). Numerical and experimental investigation on the integrate performance of axial flow cooling fan and heat exchanger. Applied Thermal Engineering, 245, 122814.

12. *ISO 5801:2017* specifies procedures for the determination of the performance of fans of all types except those designed solely for air circulation
13. ISO 3966 Measurement of fluid flow in closed conduits — Velocity area method using Pitot static tubes
14. Kaczmarzyk, P., Ziegler, B., Warguła, Ł., Burdzy, T., Popielarczyk, T., Sowa, T., & Antosiewicz, P. (2025). Experimental Studies and Computational Fluid Dynamics Simulations to Evaluate the Characteristics of the Air Velocity Profile Generated by the Positive Pressure Ventilator. *Applied Sciences*, *15*(5), 2332
15. NIST (2013). Special Publication 1019. Sixth Edition Fire Dynamics Simulator User's Guide.
16. McDermott, R., McGrattan, K., & Hostikka, S. (2008). Fire dynamics simulator (version 5) technical reference guide. *NIST Special Publication*, *1018*(5), 3-4.
17. Fire Dynamics Simulator (FDS) and Smokeview (SMV). *Natl. Inst. Stand. Technol. Gaithersburg Md.*
18. Kaczmarzyk. P.. Małozięć. D.. Burdzy. T. *et al.* Analysis of the air stream flow parameters generated by the positive pressure ventilator—full scale experiment and CFD simulation. *Sci Rep* **14**. 6852 (2024). https://doi.org/10.1038/s41598-024-57112-z
19. Fritsche, M.; Epple, P.; Delgado, A. Development of a Measurement Method for the Classification and Performance Evaluation of Positive Pressure Ventilation (PPV) Fans. In Fluids Engineering Division Summer Meeting 2018 (Vol. 51579, p. V003T12A021). American Society of Mechanical Engineers.