Grzegorz Kubicki[a) *], Izabela Tekielak-Skałka[b)]

[a)] *Warsaw University of Technology / Politechnika Warszawska*
[b)] *SMAY Sp. z o.o.*
[*] *Corresponding author / Autor korespondencyjny: grzegorz.kubicki@pw.edu.pl*

Results of Jet Fan Tests Using Experimental and Numerical Techniques

Wyniki badań wentylatorów strumieniowych z wykorzystaniem technik eksperymentalnych i numerycznych

ABSTRACT

Aim: The aim of the experiments was to check if different jet fans with similar parameters had a similar air flow profile. The study was also aimed at testing whether normal and reverse flow direction have the same profile. Next, the obtained results of velocity distribution along the airflow axis were compared with the results of numerical analysis carried out using tools commonly applied in Poland.

Project and methods: The study involved three jet fans (W1 and W3 were manufactured as reversible units, W2 was a unidirectional device). The tests were conducted in two empty warehouses to investigate airflow velocity. The measurements were performed along the axis of the fans and at additional specific points. The first measurement point was located at the fan inlet plane. The following measurements were conducted at 0.6-meter intervals at a distance of 3.6 m from the fans and at 1.2-meter intervals at a distance from 3.6 m to 24.0 m from the fan. The velocity at each measurement point was determined as the average of a 10-second measurement. The velocity measurements were conducted using an ultrasonic anemometer – Windmaster Pro. At the second stage of the study, CFD analysis was performed. Two models were devised in both Ansys Fluent and FDS. Each CFD model presented a single fan in warehouses. Models included the actual position of the fan, doors, columns and joists. Different settings and different sizes of the computational mesh were used in CFD simulation.

Results: The study resulted in an air profile along three different jet fans. Velocity profiles in normal and reverse directions were compared. Significant differences were found between airflows for normal and reverse directions. Additionally, it was possible to compare the obtained results in real scale and a CFD simulation performed in the ANSYS FLUENT 13, FDS 5.5.3 and FDS 6. Some of the CFD simulations provided a good similarity of airflow profiles in CFD and real tests, while others did not. Thus, the study showed which settings provided the best results.

Conclusions: Each of the tested fans is characterised by a different airflow distribution. The velocity distribution profile is different for the normal and reverse direction in the studied reversible fans despite the same air stream blown in both directions. The performed analyses show that CFD programs can solve velocity correctly, but this requires good settings.

Keywords: numerical simulations, velocity profile, jet fans, physical research, smoke and heat control

Type of article: original scientific article

Received: 18.10.2019; Reviewed: 19.03.2020; Accepted: 26.03.2020;
Authors` ORCID IDs: I. Tekielak-Skałka – 0000-0001-7152-8601; G. Kubicki – 0000-0001-9782-4468;
Percentage contribution: I. Tekielak-Skałka – 55%; G. Kubicki – 45%;
Please cite as: SFT Vol. 55 Issue 1, 2020, pp. 6–14, https://doi.org/10.12845/sft.55.1.2020.1;

ABSTRAKT

Cel: Celem badań było sprawdzenie, czy różne wentylatory strumieniowe o różnej konstrukcji, ale zbliżonych parametrach pracy, generowały podobny profil przepływu powietrza. Badania prowadzone były w warunkach pracy normalnej i rewersyjnej. Uzyskane rezultaty pomiarów rozkładu prędkości wzdłuż osi przepływu powietrza zostały wykorzystane do porównania wyników analiz numerycznych wykonywanych przy użyciu powszechnie stosowanych w Polsce narzędzi.

Projekt i metody: Przedmiotem badań były trzy wentylatory strumieniowe różnych producentów. Dwa z nich (W1 i W3) posiadały zdolność do pracy w kierunku normalnym i rewersyjnym, W2 był urządzeniem jednokierunkowym. Testy zostały przeprowadzone w pustej przestrzeni hali magazynowej. Pomiary prędkości przepływu powietrza, wykonane za pomocą anemometru ultradźwiękowego (WindmasterPro), realizowane były w osi wentylatorów, a pomiary wykonywane były co 0,6 m w odległości do 3,6 m od wylotu wentylatora i co 1,2 m w odległości od 3,6 do 24,0 m. Prędkość w każdym punkcie określano jako średnią z 10 s pomiaru. Badania numeryczne dla każdego z wentylatorów zostały wykonane na dwóch modelach stworzonych w Ansysy Fluent i FDS. Modele zawierały pełne odwzorowanie położenia wentylatorów oraz elementów konstrukcyjnych hali. W symulacjach zastosowano różne ustawienia i rozmiary siatki obliczeniowej.

Wyniki: Rezultatem badań było określenie i porównanie charakterystyki przepływu powietrza dla trzech różnych wentylatorów strumieniowych. Pomimo podobnej wielkości jednostek wyniki badania wykazały znaczącą różnicę między poszczególnymi charakterystykami przy przepływie normalnym i pomiędzy przepływem powietrza w kierunku normalnym i odwrotnym. Wyniki rzeczywiste zostały porównane do wyników symulacji CFD przeprowadzonych w ANSYS FLUENT 13, FDS 5.5.3 i FDS 6. Niektóre symulacje CFD pozwoliły na uzyskanie dobrego podobieństwa profilu przepływu powietrza w CFD i rzeczywistych testach, a niektóre nie. Pozwala to na określenie ustawień, przy których uzyskane wyniki najlepiej odwzorowują stan rzeczywisty.
Wnioski: Każdy z badanych wentylatorów charakteryzuje się innym rozkładem przepływu powietrza przy pracy normalnej. Znaczną różnicę widać również przy pracy badanych wentylatorów w kierunku normalnym i rewersyjnym, pomimo takiej samej wielkości strumienia powietrza wdmuchiwanego w obu kierunkach. Przeprowadzone analizy pokazują, że programy CFD potrafią poprawnie rozwiązać prędkość, ale potrzebują dobrych ustawień.
Słowa kluczowe: symulacje CFD, wentylacja strumieniowa, profil prędkości, kontrola dymu i ciepła, badania obiektowe
Typ artykułu: oryginalny artykuł naukowy

Przyjęty: 18.10.2019; Zrecenzowany: 19.03.2020; Zaakceptowany: 26.03.2020;
Identyfikatory ORCID autorów: Tekielak-Skałka – 0000-0001-7152-8601; G. Kubicki – 0000-0001-9782-4468;
Procentowy wkład merytoryczny: I. Tekielak-Skałka – 55%; G. Kubicki – 45%;
Proszę cytować: SFT Vol. 55 Issue 1, 2020, pp. 6–14, https://doi.org/10.12845/sft.55.1.2020.1;

Introduction

Jet fan systems are commonly used in ventilation systems of indoor parking lots. The basis for designing such systems is knowledge of the performance characteristics of jet fans and their accurate presentation using CFD analysis. It is important to know the air distribution formed by jet fans. This paper presents the results of physical and numerical tests of four jet fans made by different manufacturers. The objective of the study was to determine how the differences in the fans' construction and their mode of operation influence the velocity distribution model. The results were used to verify the numerical analyses of the operation of the studied fans. The CFD (Computational Fluid Dynamics) analysis was applied using: ANSYS FLUENT 13.0, FDS (Fire Dynamics Simulator), versions 5.5.3 and 6. The comparison of the results of the simulation and the survey served as a basis for the assessment of the compliance of the numerical analysis with real-life conditions.

Software

The tests described in this paper were performed by:
- Ansys Fluent 13:
  - Developer: ANSYS Inc.
  - Address: http://www.ansys.com/about-ansys/contacts-and-locations
  - Available in ANSYS Inc.: 2006
- Fire Dynamics Simulator (FDS 5.5.3 and FDS 6):
  - Developer: National Institute of Standards and Technology (NIST)
  - Address: https://www.nist.gov/services-resources/software/fds-and-smokeview
  - Year first available: 2000
  - FDS is free software
  - FDS require Smokeview to visualise the results of a simulation, also free software
  - FDS and Smokeview are available for the Windows, Linux and Mac OS X platforms.

The application of jet fans is a very popular method of day-to-day ventilation, as well as a means of smoke and heat control in enclosures like indoor parking lots. The factors influencing the efficiency of this equipment have been identified and described for years, and have been the subject of numerous surveys and analyses, the results of which are frequently published in international literature. The above-mentioned issues have been widely described, among others, in the series of articles published in the "Fire Safety Journal" [1–2], [3]. The authors of these publications agree on the fact that day-to-day ventilation systems based on the operation of jet fans, as well as smoke and heat control systems are very difficult to design using only analytical methods. At the design stage, it is almost impossible to test the assumptions using physical experiments. On the other hand, physical tests using small-scale models do not fully reflect the actual conditions of a fire. Such a thesis was presented, among others, by I. Horváth , J. van Beeck and B. Merci in a paper published in the "Fire Safety Journal" [4].

An important conclusion from study findings described in existing literature is that the velocity profile and airflow distribution along the jet fan axis is of essential for effectively directing air and smoke flow in an indoor parking lot. Such conclusions were formulated, among others, by B.J.M van Giesen., S.H.A. Penders, M.G.L.C. Loomans, P.G.S. Rutten and J.L.M. Hensen in the paper entitled *Modelling and simulation of a jet fan for controlled airflow in large enclosures* [5]. It is of paramount importance to know precisely the behaviour of the air jet generated by a fan to be able to set up jet fans in appropriate locations, and to choose properly shaped deflectors. Consequently, the location of these devices and equipping them with components that deflect the air jet is crucial in designing an effective system of day-to-day ventilation and smoke and heat control. Some countries, such as the United Arab Emirates, introduced obligatory design requirements that the manufacturer should provide full specifications of the fan's operation that included specifications of the air jet. Moreover, when designing the whole system of jet air ventilation with the use of CFD, there is a requirement to verify the behaviour of a single fan with the same CFD tools. Such verification should precede the core analysis of the whole system. The verification process should be carried out using the same tools and principles meant for a simulation in a parking lot. The obtained results of the velocity distribution field should be in compliance with the results of the physical jet fan tests (these should be the tests results provided by the equipment manufacturer). The difference between simulation results and physical tests should not exceed 10% [6]. Using this procedure, we obtain confirmation at an early stage of the design process that the used tools and methodology are able to reproduce the real-life operation of jet fan ventilation systems.

The design of jet fan ventilation systems in Poland is mostly based on the guidelines of British [7] and Dutch [8] standards, as well as on national guidelines formulated by the Building Research Institute [9]. The mentioned documentation does not include a uniform requirement that the design should be based on the specifications of the jet fans' exhaust profile provided by the manufacturers. Therefore, the design process is based on familiarity with the fan's efficiency and thrust.

At the design stage, the efficiency of the jet fan ventilation system is validated by CFD simulations. However, in Poland it is not obligatory to perform analyses according to uniform and commonly accepted guidelines, and such analyses do not have to be conducted by experts. Given these circumstances, CFD simulations are frequently superficial, using only the basic capabilities of the FDS. This leads to a situation where the performed simulations are not credible as regards the proper reproduction of real-life situations. The cases where measurements of the designed systems conducted in a real-life scale and in identical conditions as the simulations do not confirm the system's functionality derived from the numerical analysis also occur too frequently.

Practical experience warrants a thesis that the air jet generated by the fans with similar catalogue parameters may vary greatly by their velocity profile. The velocity distribution profile affected by the individual features of the design of the devices made by different manufacturers influences the air and smoke flow profile in the parking lot's cross-section. The knowledge of the behaviour of the used equipment will determine the efficiency of the ventilation and smoke and heat control systems. For the correct configuration of the network of jet fans in a parking lot, it is necessary to know the real velocity distribution profile of a specific device. The data of the actual range and the width of the air jet generated by the fan should be provided by the manufacturer. It is only by having such data at one's disposal that it would be possible to set up jet fans in correct locations within the parking lot space. This issue becomes particularly significant for the functioning of the jet fan systems in relatively small parking lots. A faulty design will not operate in accordance with the guidelines. The airflow generated by incorrectly located fans can, for example, be reflected off the wall and cause strong air and smoke circulation. In many countries, it is not compulsory for the manufacturer to provide detailed information about the air velocity distribution profile of a specific type of fan. In view of the fact that these data can influence the actual efficiency of the jet fan ventilation system, the lack of such requirement should be regarded as undesirable.

This paper presents the results of physical and numerical tests of selected types of jet fans. The studies determined the actual airflow generated by similar devices having different shapes. World literature very rarely provides the results of real-life measurements of jet fans. The data obtained in the course of the following study served as a basis for the assessment of different CFD tools.

Aim and scope of the experiment

The aim of the experiment was to test whether different jet fans with similar parameters had a similar air throw profile. The study also aimed to test whether normal and reverse flow directions have the same profile. The obtained results of velocity distribution along the airflow axis were then used to test the compliance of the results of the numerical analysis carried out using the tools commonly applied in Poland.

Methodology

Experimental setup

The study involved three jet fans. Fans W1 and W3 were manufactured as reversible units and fan W2 was a unidirectional device. Each jet fan was made of a rotor, blades and attached silencers. During the normal operation of the fan, the air first flows around the rotor and then around the engine. During the reverse operation, the air first flows around the engine and then around rotor blades. Although we can come across modern solutions with similar design in both directions, the above-mentioned property is a distinctive feature of the majority of commonly used jet fan designs.

Typical operational parameters of the fans selected for the tests are shown in Table 1.

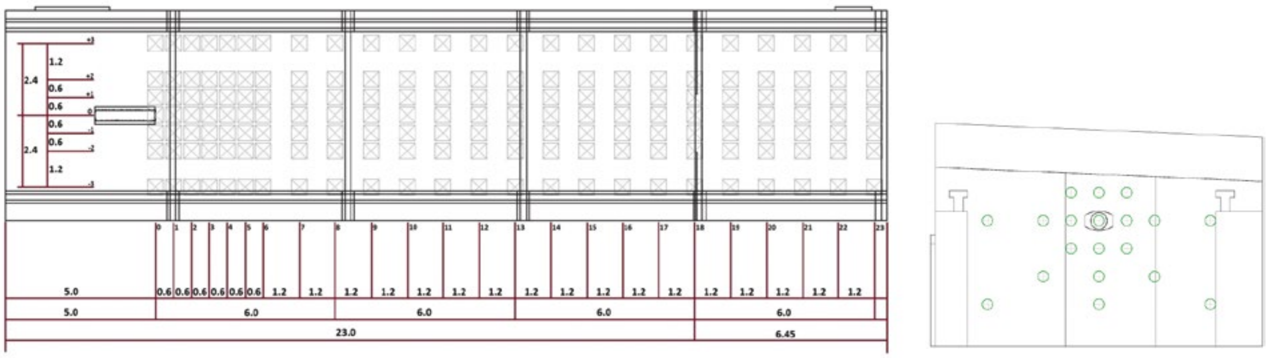
The jet fan tests were conducted in two empty warehouses located in Krakow (Poland). Two of the selected fans (W1, W2) were tested in a 7.5-metre wide and 30-metre long room, with the average height of 5 meters. The test site ensured that the air jet generated by the fans was well-isolated from the influence of the external environment. Each fan was installed at the same height. In both cases a forklift was used to lift the fan to the height of 3 meters, and this height guaranteed that the fan was able to generate a free air jet. The subsequent tests were conducted in a bigger warehouse measuring 80 by 36 meters.

The experiment tested airflow velocity. The measurements were conducted along the axis of the fans and at specific points of air jet development. The first measurement point was located at the fan exhaust plane. The following measurements were conducted at 0.6-meter intervals at a distance of 3.6 m from the fans and at 1.2-meter intervals at a distance from 3.6 m to 24.0 m from the fan. The velocity at each measurement point was determined as the average of a 10-second measurement. The test site schematic with the location of measurement points is shown in Figure 1.

Table 1. Technical data of the tested fans

Fans	Characteristics				
	Diameter, *d* [mm]	Power, *P* [kW]	Thrust, *N* [N]	Volume, *V* [m³/s]	Direction of airflow -
Fan W1	370	1.23	48	2.19	reversible
Fan W2	355	1.80	56	2.20	unidirectional
Fan W3	355	1.50	36	1.89	reversible

Source: Own elaboration.



a) b)

Figure 1. Location of measurement points in the smaller room: a) horizontal section, b) vertical section
Source: Own elaboration.

The velocity measurements were conducted using an ultrasonic anemometer – Windmaster Pro. This type of ultrasonic anemometer allows the determination of the speed of the airflow within the range of 0 to 65 m/s, with inaccuracy not exceeding 1.5% for the air speed range from 0 to 12 m/s and not exceeding 1.0% for speeds exceeding 12 m/s. It is also possible to precisely determine the airflow direction which, in consequence, allows for the exact description of the exhaust jet coming from the fan opening.

The values of the measured parameters were recorded using custom-made computer software – SAFETYCAR.

The above-mentioned site was used to test the velocity distribution profile of fans W1 and W2. The test of fan W3 was conducted in a larger room with the identical location of measurement points. Additionally, after the measurements, a tracer gas test was performed which allowed the visualisation of the operation of the two fans (fan W1 and W3).

Figure 2. View of the measurement site for testing the jet fan air flow profile
Source: Own elaboration.

Computational Analyses

At the second stage of the study, CFD analysis was performed. Models were made in both Ansys Fluent and FDS. Each CFD model included a single fan in warehouses. Models include the actual position of the fan, doors, columns and joists. Analyses were performed for a normal direction of fan W2.

The Ansys Fluent model used a tetrahedral computation grid with the maximum dimension of elements of 0.25 m. The grid element size was smaller at the walls – 0.20 m, and on the fan – 0.02 m. Basic model settings were applied. K-epsilon in the RNG version model was chosen as the turbulence model. Analysis was performed for isothermal conditions. The fan was modelled by adding two connected velocity inlet boundary conditions, one of which was blowing air and the other removing it.

Next models were created in FDS 5.5.3. The size of the computational grid was selected on the basis of the fan's diameter in such a way that the fan exhaust field in FDS had a similar size to real-life conditions. Two models were created with different mesh size: 0.15 m and 0.30 m. The jet fan was modelled as square vent with a constant volume flow of 2.2 m³/s. CFD analysis was performed for:

– the basic setting of the LES model,
– LES model with a Smagorinsky coefficient of 0.1,
– LES model with a Dynamic Smagorinsky function.

Additionally, a similar model was devised in FDS 6. A mesh was used in the model, with a piece sized 0.15 m. CFD analysis was performed for basic settings of the LES model. The jet fan in this version of FDS was modelled as a square boundary condition &HVAC with a defined diameter and constant volume flow of 2.2 m³/s.

Each simulation was performed in transient time. The results were averaged after 600 seconds for a period of 10 seconds.

Results and discussion

Results of measurements

Figure 3 present the results of velocity distribution measurements taken along the axis of the tested fans. In the case of reversible fans W1 and W3, a distinct change in their behaviour in normal and reverse modes of operation can be noted.

The shape of the velocity distribution field generated by the jet fan is as important as the velocity distribution along the fan's axis for shaping the velocity distribution profile in the parking lot cross-section. It can be surmised from the conducted measurements that each of the tested units exhibits a different velocity distribution profile.

In the case of reversible fans, the velocity distribution profile changes with the airflow direction mode. All reversible units exhibited the same phenomenon. During the normal direction mode, the highest airflow velocity was recorded along the fan's axis. When the fan operated in the reverse mode, the situation was completely different. The highest exhaust velocity was recorded near fan walls and the angle of the air throw spread also increased (Figure 4). This means that during the phase of designing smoke and heat control systems, it is necessary to perform a separate analysis for the different modes of operation of reversible fans. This requirement does not have to be upheld if the manufacturer declares an identical velocity profile in both directions of the fan's operation. Nowadays, special two-rotor jet fans are produced which, according to the manufacturers' declarations, maintain both the airflow and behaviour of the exhaust in both operational modes. In the majority of the cases, however, the data concerning the airflow in front of the fan opening are not

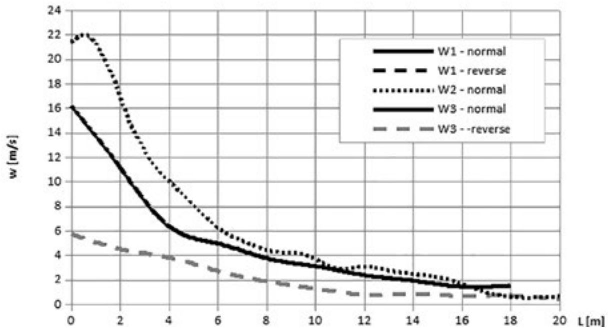


Figure 3. Comparison of air speed along the axis of fans W1; W2 and W3 operating in the normal direction

Source: Own elaboration.

Table 2. Test of air velocity produced by fan W2 at a fan height at different points shown in Figure 1

L [m]	-3	-2	-1	0	+1	+2	+3
0.0	0.29	0.28	0.30	21.32	0.14	0.19	0.11
0.6	0.37	0.21	0.48	21.98	0.34	0.16	0.12
1.2	0.11	0.42	0.41	20.60	0.38	0.18	0.13
1.8	0.32	0.10	0.43	18.10	0.29	0.24	0.27
2.4	0.28	0.24	1.38	14.81	0.46	0.20	0.15
3.0	0.13	0.22	2.20	12.62	0.77	0.61	0.21
3.6	0.29	0.27	2.86	10.87	1.33	0.36	0.28
4.8	0.12	0.51	4.34	8.55	1.60	0.39	0.17
6.0	0.20	1.26	3.95	6.27	1.86	0.61	0.36
7.2	0.20	1.34	4.52	5.02	1.71	0.49	0.62
8.4	0.32	2.18	4.51	4.26	1.40	0.41	0.62
9.6	0.24	2.39	4.03	4.02	1.82	1.02	0.54
10.8	0.68	1.91	3.09	2.96	2.16	0.99	0.60
12.0	0.46	2.35	3.13	3.05	2.00	0.90	0.88
13.2	1.19	2.93	2.63	2.69	2.04	1.48	0.91
14.4	0.88	2.05	2.21	2.38	1.81	1.19	0.37
15.6	0.48	1.24	0.84	1.93	1.26	0.99	0.58
16.8	0.38	0.66	1.01	1.08	1.24	0.41	0.37
18.0	xxx	0.38	1.17	0.60	0.55	xxx	xxx
19.2	0.30	0.34	0.46	0.56	0.65	0.14	0.11
20.4	0.30	0.34	0.25	0.65	0.74	0.22	0.11
21.6	0.33	0.19	0.18	0.47	0.51	0.40	0.18
22.8	0.39	0.16	0.14	0.34	0.55	0.36	0.15
24.0	0.15	0.46	0.35	0.23	0.12	0.12	0.33

Source: Own elaboration.

provided. The results of the tests conducted, among others, by the authors of this paper reveal that the behaviour is of a completely different nature. We believe that the information concerning the method of the air discharge from the fan is essential for the efficiency of the whole system of parking lot ventilation.

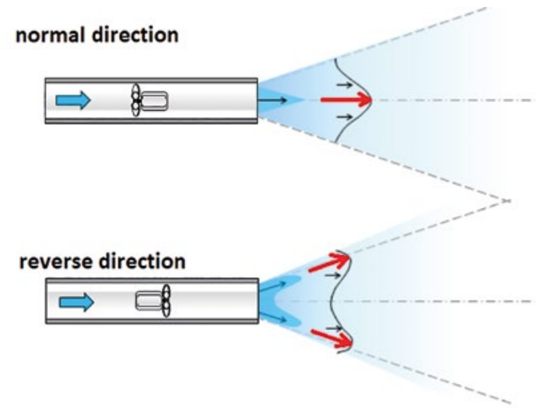


Figure 4. A comparison of the air throw performance of different jet fans
Source: Own elaboration.

Comparison of study results with the CFD analysis using different tools

Results of CFD simulations

The above-mentioned physical experiment provided an excellent opportunity to verify the precision of the commonly used models of numerical analyses. It was possible to compare the results obtained from the ANSYS FLUENT 13 program and FDS 5.5.3 and FDS 6 (commonly used in Poland) with the measurement results of fan operating parameters. Below please find the selected results of a W2 fan simulation.

Firstly, the measured velocity along the jet fan axis was compared with velocity distribution obtained from the simulation performed in the ANSYS FLUENT 13 program.

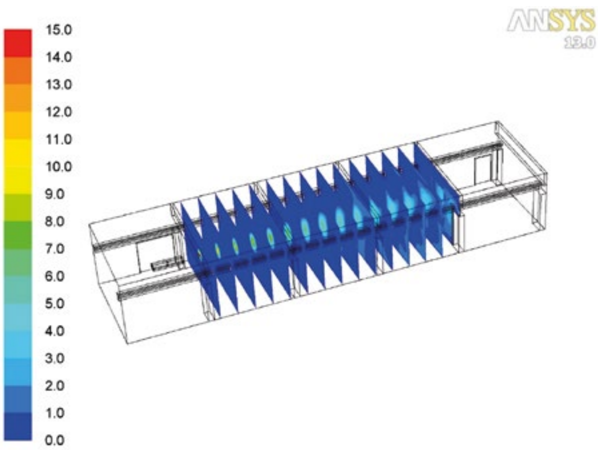


Figure 5. Visualisation of velocity distribution (m/s) in measurement cross-sections in the Ansys Fluent program
Source: Own elaboration.

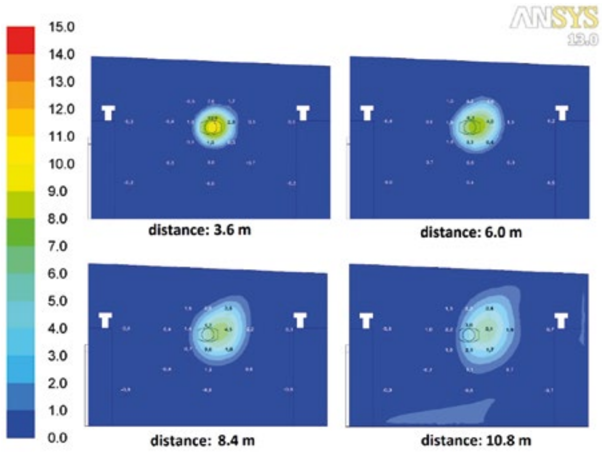


Figure 6. Results of the simulation of velocity (m/s) in Ansys Fluent at different distances from the fan
Source: Own elaboration.

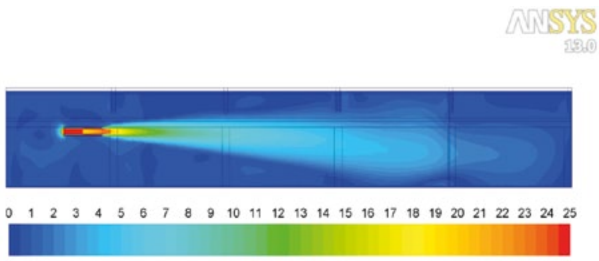


Figure 7. Visualisation of speed distribution in the Ansys Fluent
Source: Own elaboration.

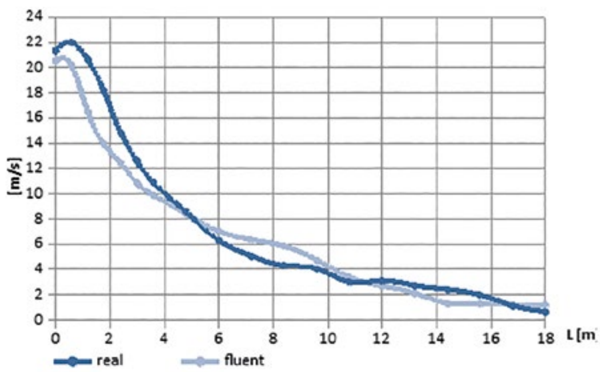


Figure 8. Comparison of the measured velocity profile of fan W2 with the results obtained in the Ansys Fluent program
Source: Own elaboration.

The basic settings of the Ansys Fluent program, in combination with a dense computational grid, yielded a satisfactory reproduction of the exhaust velocity distribution of the studied fan. Unfortunately, due to the high costs of a commercial license, this program is very rarely used in Poland for numerical analyses of indoor parking lots.

Another figure compared the measurement and numerical velocity along the fan's axis with the velocity distribution simulated by FDS v 5.5.3.

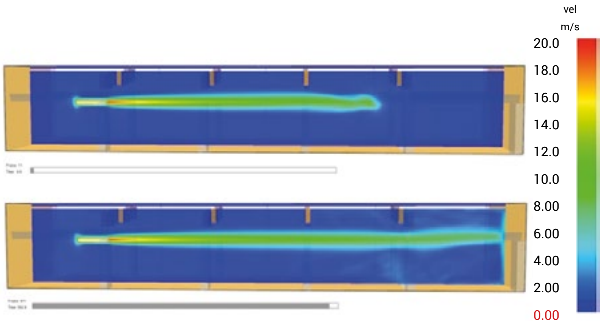


Figure 9. Visualisation of velocity distribution (m/s) in the program FDS 5.5.3 for the default settings of the LES model and a 15-cm grid (after 10 and 600 seconds)

Source: Own elaboration.

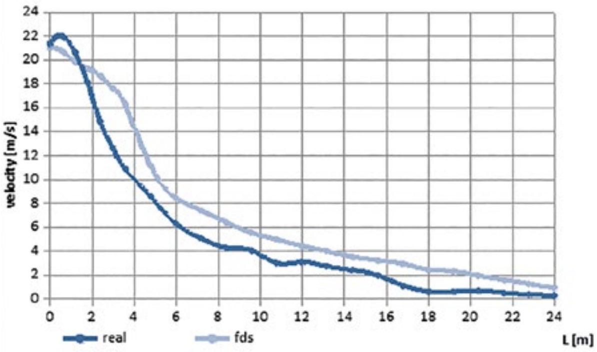


Figure 12. Velocity comparison measured along the jet fan's axis with velocity distribution simulated in the FDS 5.5.3 program with the use of the constant value of the Smagorinsky coefficient, equal to 0.1, and a 15-cm grid

Source: Own elaboration.

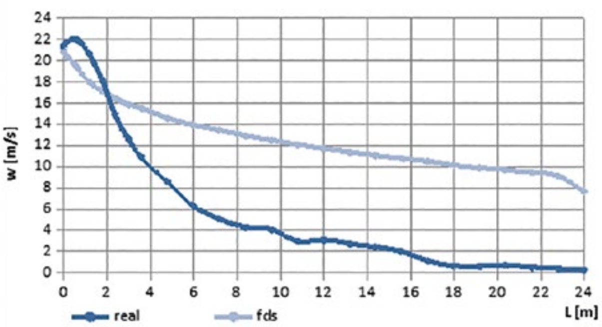


Figure 10. Velocity comparison measured along the fan's axis with velocity distribution simulated in the FDS 5.5.3 program in the LES model and grid settings of 15 cm

Source: Own elaboration.

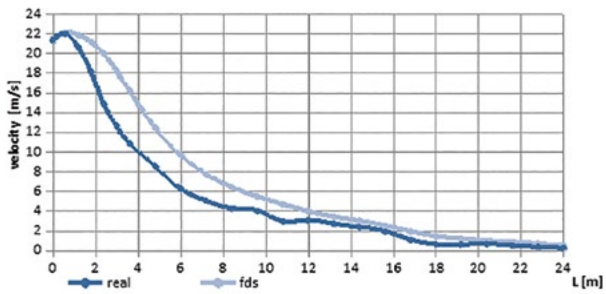


Figure 13. Velocity comparison measured along the jet fan's axis with velocity distribution simulated in the FDS 5.5.3 program with the use of the Dynamic Smagorinsky function, and a 15-cm grid

Source: Own elaboration.

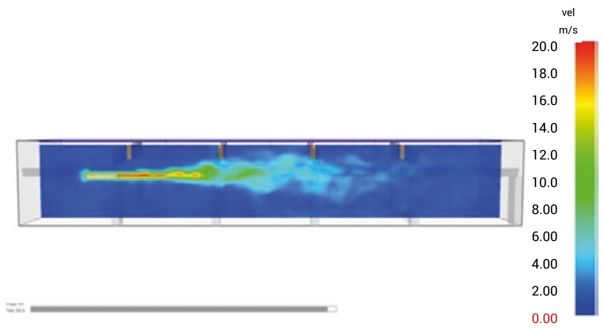


Figure 11. Visualisation of velocity distribution (m/s) in program FDS 5.5.3 with the use of the Dynamic Smagorinsky function in the LES model with a 15-cm grid

Source: Own elaboration.

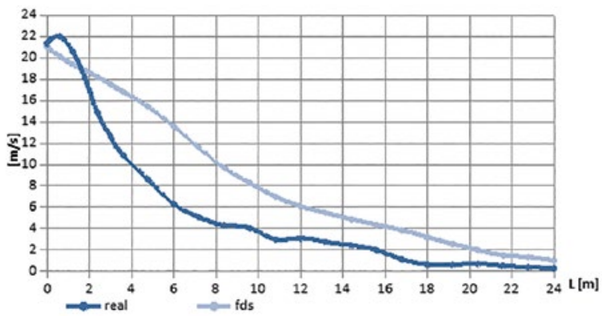


Figure 14. Velocity comparison measured along the jet fan's axis with the velocity distribution simulated in the FDS 5.5.3 program with the use of the Dynamic Smagorinsky function in the LES model for a 30-cm grid

Source: Own elaboration.

The simulation results illustrated by Figures 12, 13 and 14 reveal that it is necessary to use the Dynamic Smagorinsky function properly in the FDS program version 5.5.3. The impact of the density of the computational grid on the precision of the simulation is also clearly seen.

In conclusion, the same analysis was performed in the newer – sixth – version of the FDS program. The results obtained for the basic program settings are presented in Figure 15. The simulation results revealed that immediately outside the fan exhaust opening, the values were lower than anticipated; however, farther on,

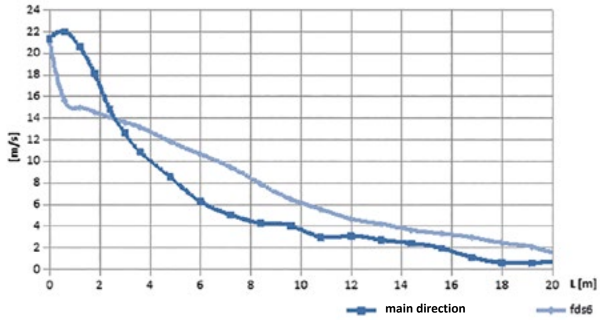


Figure 15. Comparison of velocity measured along the jet fan's axis with velocity distribution obtained in the FDS 6v program with basic program settings

Source: Own elaboration.

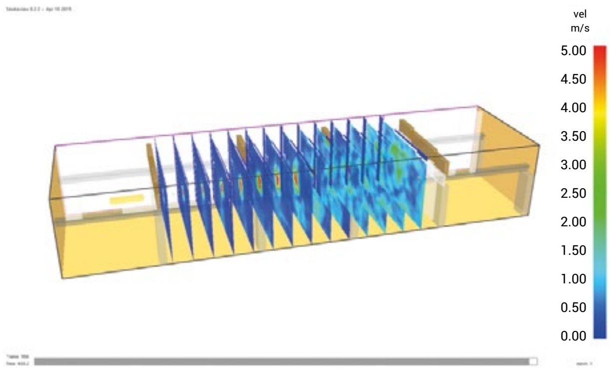


Figure 16. Visualisation of velocity distribution (m/s) at cross-sections over 600 seconds in the FDS 6 program with basic program settings for a 15-cm grid

Source: Own elaboration.

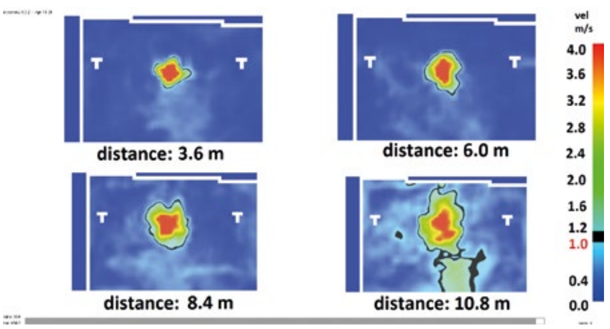


Figure 17. Visualisation of velocity distribution (m/s) in FDS 6 at different distances from the fan

Source: Own elaboration.

the simulation results showed overstated values in comparison with the measurements.

Conclusions

The results of the study:

1. Each of the tested fans exhibits a different airflow distribution. In the case of the forward direction, the angle of airflow spread for different fans ranged from 18 to 30 degrees. Fans W1 and W2 featured small air stream diffusion at the exhaust opening, making it possible to maintain high velocity along the fan's axis at a long distance. Fan W3 showed a lower velocity along the fan's axis and the airflow was more diffuse and curved.
2. The velocity distribution profile is different between the normal and reverse direction in the studied reversible fans despite the same air stream pushed in both directions. In the case of the normal direction (when air flows around the rotor first and then around the engine), the highest velocity occurs along the fan's axis. In the case of the reverse direction, the axis speed is lower by approximately a half in comparison with the basic direction. On the other hand, in the case of reversible flow, velocity is higher at fan walls.
3. The popular guidelines for conducting computer analyses to determine the minimum requirements of the computational grid (e.g., not bigger than 0.4 m x 0.4 m x 0.4 m; whereas in the fire zone and adjacent zones, the computational grid cannot be larger than 0.2 m x 0.2 m x 0.2 m) may turn out to be insufficient for the appropriate reproduction of real-life conditions. It is necessary to ensure higher precision in the determination of the requirements for simulations in relation to the guidelines for a specific CFD program. The maximum size of the computational grid should also be determined more precisely, particularly when establishing the behaviour of air spread right outside the jet fan exhaust opening. The conducted analyses show that for the proper reproduction of the air exhaust outside the jet fan opening with the use of the FDS program, the fan exhaust field should be covered by at least four cells of the computational grid, which in this case provide to be a grid with a 15-cm side.
4. Numerical analysis results showed that the results obtained depended largely on the version of FDS used. The newer version (FDS 6) yielded similar results for measurements already in the basic version of the program.

Acknowledgements

We would like to thank the SMAY Company (Poland) for the financial support of our study.

Literature

[1] Deckers X., Haga S., Tilley N., Merci B., *Smoke control in case of fire in a large car park: CFD simulations of full-scale configurations*, "Fire Safety Journal" 2013, 57, 22–34, https://doi.org/10.1016/j.firesaf.2012.02.005.

[2] Deckers X., Haga S., Sette B., Merci B., *Smoke control in case of fire in a large car park: Full-scale experiments*, „Fire Safety Journal" 2013, 57, 13, 11–21, https://doi.org/10.1016/j.firesaf.2012.10.017.

[3] Merci B., Shipp M., *Smoke and heat control for fires in large car parks: Lessons learnt from research?*, "Fire Safety Journal" 2013, 57, 3–10.

[4] Horváth I., v.d.Beeck J, Merci B., *Full-scale and reduced--scale tests on smoke movement in case of car park fire*, „Fire Safety Journal" 2013, 57, 35–43, https://doi.org/10.1016/j.firesaf.2012.10.009

[5] Van Giesen B. J. M., Penders S. H. A., Loomans M. G. L. C., Rutten P. G. S., Hensen J. L. M., *Modelling and simulation of a jet fan for controlled airflow in large enclosures*, "Environmental Modelling & Software" 2011, 26, 2, 191–200, https://doi.org/10.1016/j.envsoft.2010.07.008.

[6] UAE FIRE AND LIFE SAFETY CODE OF PRACTICE.

[7] BS 7346-7:2006 Components for smoke and heat control systems – Part 7: Code of practice on functional recommendations and calculation methods for smoke and heat control systems for covered.

[8] NBN S 21-208-2 Protection incendie dans les batiments. Conception des systems d'evacuation des fumees et de la chaleur (EFC) des parkings interieurs.

[9] Węgrzyński W., Krajewski G., *Systemy wentylacji pożarowej garaży – projektowanie ocena odbiór*, ITB Warszawa 2015.

GRZEGORZ KUBICKI, PH.D. ENG. – a specialist in the field of fire ventilation systems. For 15 years, lecturer of the fire ventilation subject and in post-graduate studies 'Building smoke extraction systems – fire ventilation'. Participant of research programmes and co-creator of technical solutions for modern smoke prevention systems for the evacuation and smoke extraction of stairwells.

DR INŻ. GRZEGORZ KUBICKI – specjalista w dziedzinie wentylacji pożarowej. Od 15 lat prowadzi na uczelni zajęcia z wentylacji pożarowej oraz wykłady w ramach studiów podyplomowych z zakresu systemów oddymiania budynków – wentylacji pożarowej. Uczestnik programów badawczych oraz współtwórca rozwiązań technicznych na potrzeby współczesnych systemów zapobiegania rozprzestrzenianiu dymu w procesie ewakuacji i oddymiania klatek schodowych.

IZABELA TEKIELAK-SKAŁKA, M.SC. ENG. – graduated from the Cracow Technical University in 2010 with a M.Sc. degree. Head of the department of research and CFD analysis at SMAY Sp. z o.o. She has been researching various fire ventilation systems for eight years.

MGR INŻ. IZABELA TEKIELAK-SKAŁKA – w 2010 r. ukończyła studia na Politechnice Krakowskiej, uzyskując tytuł magistra. Kierownik działu badań i analiz CFD w przedsiębiorstwie SMAY Sp. z o.o. Od ośmiu lat prowadzi badania nad różnymi systemami wentylacji pożarowej