Numerical study of radial high-pressure blower with frequency converter

Cite as: AIP Conference Proceedings **2449**, 050001 (2022); https://doi.org/10.1063/5.0090994
Published Online: 01 September 2022

Viktor Stoyanov and Valyo Nikolov

Numerical Study of Radial High-Pressure Blower with Frequency Converter

Viktor Stoyanov [a] and Valyo Nikolov [b]

Department of Transport and Aviation Equipment and Technologies,
Technical University of Sofia, Plovdiv Branch,
25 Tsanko Dyustabanov str., 4000 Plovdiv, Bulgaria.

[a] *Corresponding author: viktor.stoyanov@gmx.de*
[b] *vnikolov@tu-plovdiv.bg*

Abstract. Radial high-pressure air blowers are used when it is necessary to transfer air mass from one area to another accompanied by generation of significant pressure in a short period of time. In practice, this is used in case of need for rapid extraction of contaminated air or in general in the industry for intensive blowing or sucking air from or in rooms with special requirements for air quality in them. The aim of this work is to consider the specific geometry of a radial high-pressure air blower, model S-HP 470/35 - 105/20 from Elektror airsystem Gmbh, for which detailed measurements according to DIN EN ISO 5801 Fans - Performance testing are made. As a result of the measurements made for this particular blower, a detailed ventilation characteristic curve is created. The selected range for validation of the numerical results compared to the experimental measurements coincides with the range of ventilation characteristic curve from 0m^3/min to 48.2 m^3/min and pressure increases up to 20,000 Pa and accelerates up to 105 Hz (approx. 6,200 rpm) on the blower's impeller. For this purpose, a 3D engineering simulation model known as Computational Fluid Dynamics (CFD) and a digital twin of the real model selected for study are used. The simulations are performed in the commercial software product Star-CCM+ version 6.04.014 using the v^2f Low-Reynolds Number k-ε turbulence model. Finally, the numerical results are compared with the DIN EN ISO 5801 experimental measurements.

INTRODUCTION

What fans, blowers or compressors have in common when working with fluids is that these devices increase the air pressure of the fluid passing through them at the expense of mechanical energy expenditure on its operation [1]. Increasing the pressure in radial type devices is achieved by means of a part with special geometry (impeller) rotating around its own axis. As a result of the impeller operation the desired centrifugal effects in the fluid are caused. The classification of a device as belonging to the group of fans, blowers and compressors is made by The American Society of Mechanical Engineers (ASME) PTC 11 - 2008 (R2018) Page: 176. In general, it can be noted that the classification proposed by ASME is based on the ratio of the pressure to the suction pressure of the devices. Within this work, the classification proposed by ASME is accepted and based on this, it is defined that the object of study by using numerical simulations and experimental values is a radial pressure blower. A number of blower manufacturers make further dividing into three sub-categories: Low, Medium and High-Pressure Blowers, depending on the volume flow rate of the blowers. The blower S-HP 470/35 - 105/20 from Elektror airsystem Gmbh studied in this work is characterized by pressure increase up to 20 000 Pa at acceleration up to 105 Hz (approx. 6,200 rpm) on the blower's impeller and maximum volume flow rate of 95 m^3/h. According to its own methodology the manufacturer classifies this blower as a radial high-pressure blower.

10th International Scientific Conference "TechSys 2021" – Engineering, Technologies and Systems
AIP Conf. Proc. 2449, 050001-1–050001-10; https://doi.org/10.1063/5.0090994
Published by AIP Publishing. 978-0-7354-4397-6/$30.00

There are a number of fundamental scientific papers that describe the basic principles of operation of fans, blowers or compressors and the corresponding mathematical models that allow preliminary analytical calculations when designing them [1, 4, 6]. In his scientific works R. K. Turton [1] described methods for practical solution of the problem of designing fans, blowers and compressors in the 80s of the last century. He also published brief summaries of some of his work [5] and later in 1994 revised and updated his book Principles of Turbomachinery [2]. In his book published in 1984, R. K. Turton [1] presented a fundamental "black box" model of a machine operating according to hydrodynamic principles described using Euler's equations. This model is gradually being upgraded and the same author offers a number of theoretical mathematical models for designing fans, blowers and compressors. S. Korpela [3, 4] also worked in this direction and published a number of scientific papers which describe the principle of operation of fans, blowers and compressors and enabling the theoretical solution of specific examples from practice.

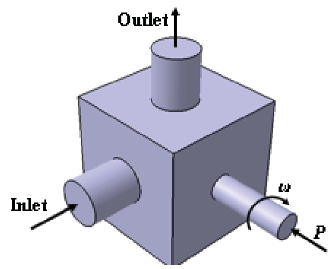

FIGURE 1. Blowers "black box" [1].

As a result of the constantly rising requirements towards blowers, concerning not only their functional reliability but also their energy efficiency, the past conventional method of designing blowers which includes trial-and-error empirical methods significantly limits the ability to achieve high accuracy in practice. This is due to the simplifications made regarding empirical coefficients for determining the losses due to impact and clearance losses in the blower, including impeller and volute casing, friction, as well as deviations of air distribution between the blade channels of blowers. For a new blower's design, or newly defined operating mode, the real losses compared to the theoretical values can be determined by conducting experiments in real operating conditions. In practice, the differences between the theoretical and actual values of a ventilation characteristic curve can be significant - Figure 2 [7].

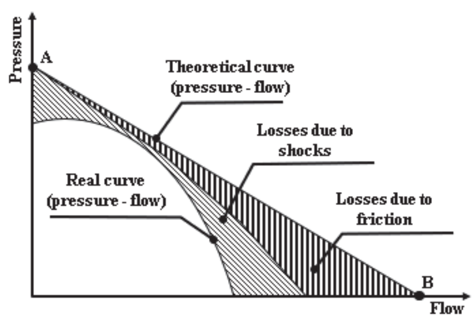

FIGURE 2. Empirical prediction of the ventilation characteristic curve [7].

This disadvantage, namely, the use of empirical coefficients that only partially describe the real physical phenomena of 3-dimensional airflows in creating new models of blowers or generally of radial high-pressure blowers requires the use of new approaches in their design.

Newer approache to blowers design which builds on and develops the expertise accumulated from empirical and analytical models, but at the same time allows to study in detail the behavior of 3D air flows in blowers, uses specialized 3D software for computational fluid dynamics (CFD). In this approach, for designing modern industrial blowers, digital twins which describe in detail the real geometry and performance characteristics of the future blowers are used. At the same time, numerical modeling describes not only the mechanical characteristics of blowers, but also the physical characteristics of the environment. Similar CFD study of centrifugal fan flow was done by Siwek T et all [8] using ANSYS CFX package. Sheard, A et all [9] used an aerodynamic and aero-acoustic virtual prototype for his research with commercial ANSYS Workbench 8.0 software, using not only CFD models to study the distribution of airflow in a centrifugal fan. Galerkin Y and Marenina L [10] also used ANSYS CFX 14 in the investigation and perfection of a centrifugal compressor. Khelladi et all [11] used a FLUENT simulation code with the Shear Stress Transport (SST) turbulent model. As maintained by the author, the obtained results are with good accuracy.

A key point in the research is the validation of the fluid flow simulation method by comparing the results of the simulation modeling with those of the experimental measurements. Because of the availability of different industrial CFD software products, as well as the lack of a unified methodology for use of numerical methods in modeling blowers through identical digital twins, deviations of the numerical results from the experimental measurements are possible. The differences in the results can be due both to the choice of turbulent models in the numerical simulations and to the type of digital twin network, but also to the choice of CFD software itself. Similar analyzes of results with different turbulent codes from identical industrial CFD codes suppliers have been made by V. Stoyanov, V. Nikolov and M. Garcia [12], who used Star-CCM+ software, Shear Stress Transport (SST), and v2f low Re k-ε turbulent models for his research. Petukhov et al [13] analyzed two turbulent models with digital twins of vane diffusers of a centrifugal compressor, respectively standard turbulence models k-ε and k-ω Shear Stress Transport (SST) integrated in Ansys CFX software package. A number of studies based on the CFD method analyze individual components of blowers and make improvements such as changing the geometry of the blades or changing the geometry of the blowers' outlet or inlet. In other studies, simulation of the whole blower is performed, and analytical and simulation results are compared with experimental results obtained in specially prepared laboratory conditions provided by various scientific organizations. In fact, there are internationally accepted standards such as DIN EN ISO 5801 Fans - Performance testing, which are used to determine the characteristics of industrial fans and blowers. In this regard, in the overall assessment of the blower performance described by means of ventilation characteristic curves, using a complex digital twin of the experiment which consists of the geometry of the blower itself and the overall experimental setup would be appropriate. DIN EN ISO 5801 Fans - Performance is an internationally accepted standard for the experimental validation of a Fan characteristics. Similar standardized experimental setups have been installed in a number of manufacturers of fans, blowers, and compressors.

The purpose of this work is:

- Creating a digital twin of a real test section which meets the requirements of DIN EN ISO 5801 Fans - Performance with an integrated digital twin of a real radial high-pressure air blower, model S-HP 470/35 - 105/20 from Elektror airsystem gmbh.
- Recording the ventilation characteristic curve of S-HP 470/35 - 105/20 to validate the numerical results with those of the experimental measurements using 3 dimensional CFD simulations with Star-CCM +.

SELECTION OF TEST CASES

The validation of the numerical results obtained through Star-CCM+ in relation to the experimental results from the air-measuring section according to DIN EN ISO 5801 is performed by direct comparing the values of the numerical and experimental ventilation characteristic curve. During the experimental measurements, the requirements described in DIN 24163 and EN ISO 516 regarding the Flow Rate measurement methodology were also considered. The experimental ventilation characteristic curve was taken in operating conditions for multiple operating points (OP), while the numerical characteristic curve was extrapolated from the results obtained for 4 OP.

TABLE 1. S-HP 470/35 - 105/20 essential geometrical parameters.

Description	Value
Impeller	
Blade number	9
Blade type	spiral
Blade thickness	2mm
Impeller width	35mm
Inlet diameter	155mm
Outlet diameter	470mm
Volute casing	
Inlet diameter	148mm
Outlet size	96x160mm

To determine the experimental ventilation characteristic curve of a radial high-pressure air blower, S-HP 470/35 - 105/20 from Elektror airsystems Gmbh with essential geometrical parameters presented in Table 1, the air-measuring test section according to DIN EN ISO 5801from Elektror airsystems Gmbh is used - Figure 3.

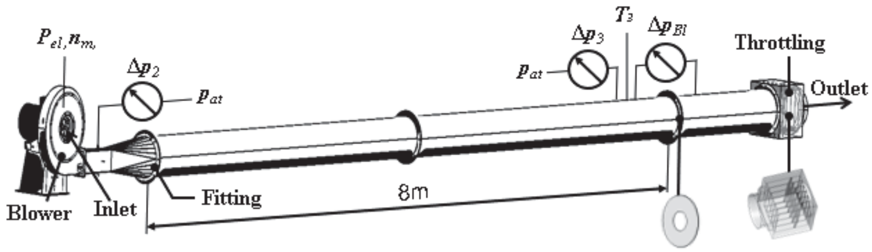

FIGURE 3. Shematic of the air-measuring test section with integrated S-HP 470/35-105/20 radial high-pressure air blowers.

The air-measuring test section is designed on the basis of DIN EN ISO 5801 and composed of an electric motor with a defined power P_{el}. In the process of study, rotational movement of the blower moving part, called impeller in the literature is transmitted using the electric motor whose operation is controlled by means of a shaft, the revolutions of which are measured. The structure of S-HP 470/35-105/20 geometry is shown in Figure 4 by using a 3D model.

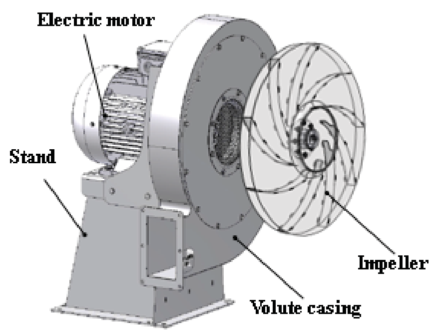

FIGURE 4. Three-dimensional geometry of S-HP 470/35-105/20.

As a result of the impeller operation, the desired centrifugal effects in the fluid are caused and thus air from the the blower surrounding through a specially designed for this purpose inlet in the volute casing of the blower is sucked in. The intake air is directed to the air-measuring test section by means of an adapter at the blowers' outlet and of a fitting. The length of the air-measuring test section is 8000 mm with a diameter of 300 mm. In order to indirectly determine the value of the volumetric flow rate required to determine the values of ventilation characteristic curve, an orifice metal plate with a diameter of 144 mm was installed. Thus, and according to the requirements of EN ISO 5167, the actual value of the volumetric flow rate relative to the specific operating mode of the blower can be calculated by measuring the pressure differences before and after the orifice, as well as considering the atmospheric pressure and temperature. This study is not intended to perform a detailed analysis of the analytical method for determining the volumetric flow using the above-mentioned methodology and physical values in the air-measuring test section. There are a number of in-depth analyzes on this subject, therefore in this study known and proved scientific methods for determining volumetric flow, based on volume flow measurement according to DIN 24163 and EN ISO 5167 are used. At the end of the air-measuring test section an additional throttling section, with the ability to work in different modes is integrated. The need to integrate these sections is due to the fact that the blower model, consisting of an impeller and volute casing, allows only two operating points (start and end) of the ventilation characteristic curve to be recorded. The initial operating point OP1 represents the operating state of the system when the blower's outlet is closed. The other case corresponding to the maximum flow through the blower is at the endpoint of the ventilation characteristic curve and it is possible to realize when atmospheric pressure is registered at the outlet of the blower. To establish intermediate values of the ventilation characteristic curve, a throttling section is installed between the fully closed or open outlet of the system as part of the air-measuring test section. Thus, the study of the blower characteristics in the entire operating range, by means of a partially controlled opening of the test section outlet it is possible.

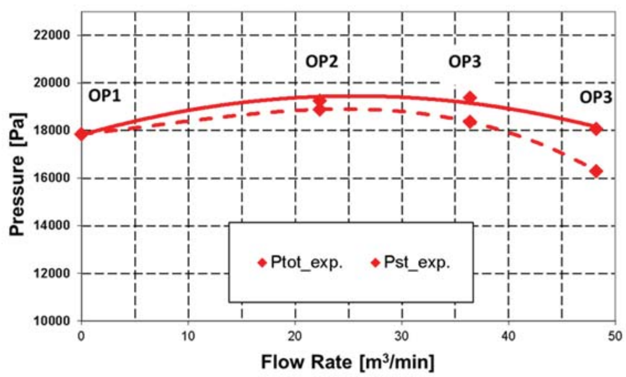

FIGURE 5. Ventilation characteristic curves of S-HP 470/35-105/20.

In this work, the blower ventilation characteristic curves were taken in operating conditions in the range of flow rate from 0m^3/min to 48.2 m^3/min, pressure increases up to 20 000 Pa at accelerates up to 105 Hz (approx. nominal 6,200 rpm) on the impeller. The experimentally obtained values of ventilation characteristic curves of S-HP 470/35-105/20 regarding static P_{st} and total P_{tot} pressure are visualized in Figure 5.

COMPUTATIONAL DETAILS

For the purposes of this work, commercial software code STAR-CCM + 6.04.014 is used. A digital twin of radial high-pressure blower S-HP 470/35 - 105/20 and an air-measuring test section DIN EN ISO 580 were modeled using digital three-dimensional computer-aided design (CAD) software - illustrated in Figure 3. In the current study of a radial high-pressure blower, a comparative analysis of the numerical and experimental values of the ventilation characteristic curves is performed.

Domain and Boundary Conditions

In this study, four simulations were performed under identical limit and initial conditions which correspond to pre-selected operating points (OPs) from the empirically obtained values of ventilation characteristic curves for total and static pressure. The difference of the digital twins corresponds to the geometry of the air-measuring test section in the individual OPs and in particular in the area of the throttle section, which is fully open at OP1 and partially closed at three different throttle positions, OP2, OP3 and OP4, respectively. During the individual numerical simulations, the initial conditions of the digital twin do not change. The simulated values of the pressure differences and the volume flow rate were reported only after a quasi-stable equilibrium state of the interaction values had been reached and at least 9000 iteration steps had been completed. Preliminary studies of the numerical model with digital twins used in this study show that after 9000 iterations, negligible fluctuations in the results have been observed. In this study the v2f Low-Reynolds Number k-ε turbulence model is used. The choice of this turbulence model is based on analyzes of results from STAR-CCM + studies in the field of impinging gas jets [12].

The digital twin of the air-measuring test section with a blower presented in Figure 6 serves for subsequent numerical studies within this study and is characterized by the following domain and boundary conditions. The working medium in the system is air at an initial state corresponding to a static pressure p_u=101325 Pa and temperature T_u=293,15K. Nominal revolutions of the Impeller corresponding to a frequency of 105 Hz for OP4 are defined at n_4=6240 rpm with a partially open orifice of the air-measuring test section. Due to its specific characteristics, the impeller area is defined as a rotating area with a rotation axis the impeller axis itself. In addition, all solid surfaces were defined as adiabatic and with a roughness of 0,1 mm.

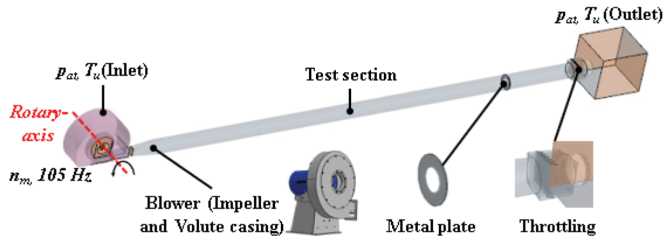

FIGURE 6. 3D Numerical modeling and boundary conditions of the digital twin.

To the already defined initial requirements of the digital twin from fig. 6 the respective boundary areas with their physical characteristics, corresponding to the experimental environment, respectively Wall, Motion, Stagnation Inlet and Pressure outlet are also determined (see Figure 7).

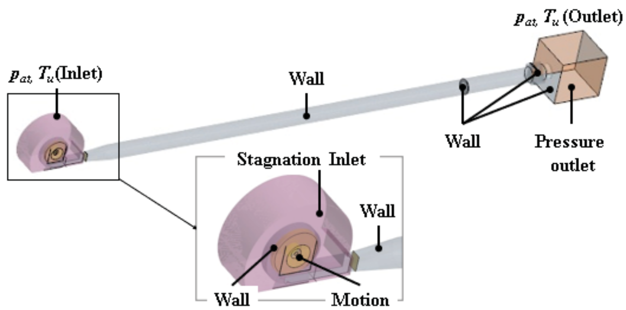

FIGURE 7. Boundary areas of the digital twin.

Grid Generator

Computational meshes are generated by using STAR-CCM+ version 6.04.014. To achieve the goals of this study, two types of cells were used, Trimmer and Polyhedral, respectively. The decision to use two types of cells in this study is based on the fact that in the study area and in particular in the blower, physical phenomena due to the rotation of the impeller around its own axis are observed. Then the air flow passes through a pipe at a ratio of length L to diameter D or L/D=266,6. Previous investigations by the author [12] have shown that the numerical results obtained in studies of air flow propagation through a pipe when using Trimmer meshing, have satisfactory accuracy. In the case of complex airflows, as observed in the impeller field, the use of Trimmer cells can lead to additional numerical inaccuracies that will propagate in the numerical range of the air-measuring section and would affect the accuracy of the final result. Prism Layer Mesher is used for both types of meshes. Further information on mesh characteristics is presented in Table 2.

TABLE 2. Main parameters of the retained grids.

Fluid domain	Surface mesh	Mean sice (mm)	Number of mesh
Inlet area	Polyhedral	4	3 575 868
Impeller	Polyhedral	2	780 273
Volute casing	Polyhedral	2	1 906 026
Test section	Trimmer	8	39 258 489

To make individual settings of the mesh parameters in the computational area (Fig. 8 - “Test section”) several additional areas are defined (Fig. 8 - “Throttle fine”, “Throttle middle” and “Plate local”). The selection of local areas for individual meshing is based on previous research on impinging gas jets [12] and contributes to improvement of the obtained numerical results accuracy.

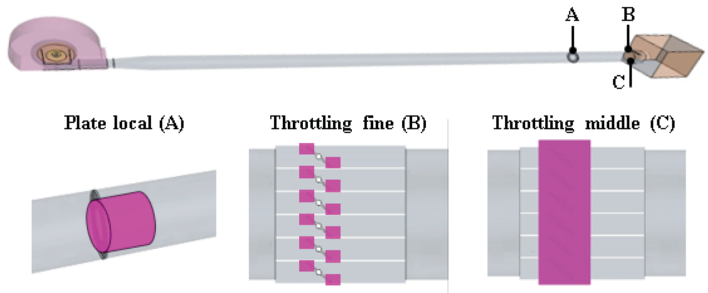

FIGURE 8. Local areas for meshing.

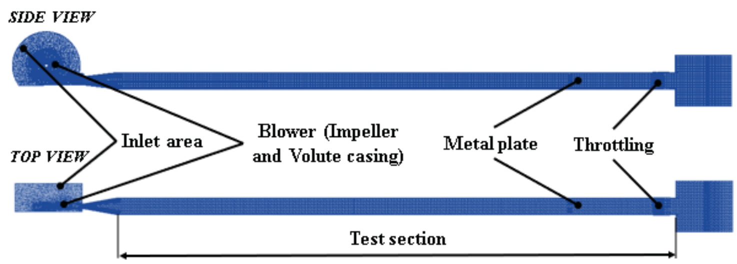

FIGURE 9. Three-dimentional view on a grid for the study of radial high-pressure blowers.

After selecting a specific length of the base cell, complete meshing of the digital twin is performed, and the meshing result which includes a total of 39,258,489 cells is presented in figure 9.

NUMERICAL PREDICTION

The numerical ventilation characteristic curve of a radial high-pressure air blower model S-HP 470/35-105/20 is defined using the local values of pressure (static and total) and flow rate in OP1 to OP4 at four valve positions of the throttle mechanism. The simulations were performed with the commercial software product Star-CCM+ version 6.04.014, using the v^2f Low-Reynolds Number k-ε turbulence model. This software offers the ability to import a completed CAD model and perform the full simulation chain which consists of pre-processor, solver and postprocessor, without the need to use other auxiliary software. To limit the influence of numerical errors in the validation of the results obtained from the simulation models, preliminary mesh independence analyze was performed. Digital twins of S-HP 470/35-105/20 and air-measuring test section under identical domain and boundary conditions were used. The impeller was modeled as a rotating body in a fixed housing. The impeller rotation speed is defined as a variable parameter with frequency converter corresponding to 105Hz. The final model was meshed with around 40 million cells. The results are validated using locally determined values of ventilation characteristic curve for ptot and pst obtained experimentally at four operating points, and their numerical values are shown in Figure 10.

Figure 10 also shows the curves interpolated from the measurements and numerical simulations of the ventilation characteristic curve in the range from $0m^3$/min to 48,2 m^3/min.

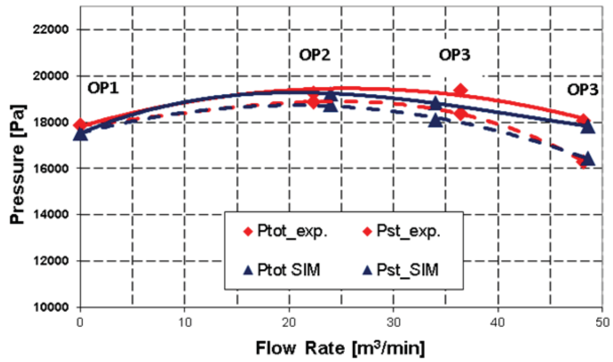

FIGURE 10. Experimental and simulated ventilation characteristic curves (static and total pressure).

Visualization of the velocity distribution in the blower as a result of the rotated impeller and in general in the digital twin of the air-measuring section is presented with a scalar scene in Figure 11.

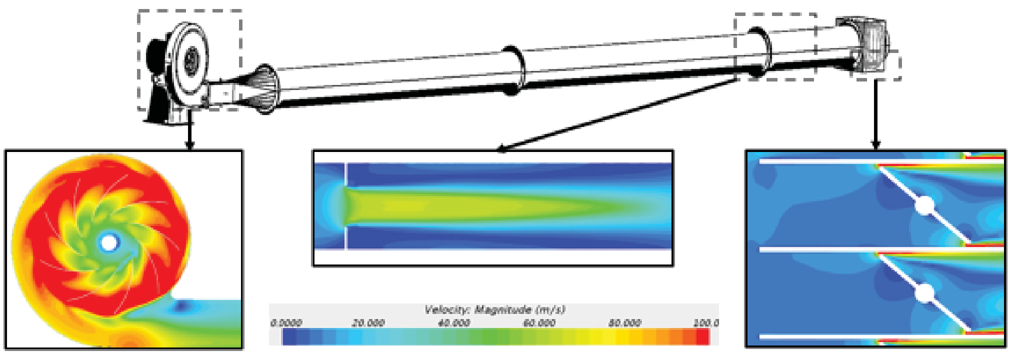

FIGURE 11. Speed distribution in the Virtual air-measuring section.

The impeller of radial high-pressure air blower, model S-HP 470/35 - 105/20 has a spiral shape of the ribs. At the same time, the impeller is not centrally located in the casing, which is typical for blowers and this leads to an inhomogeneous distribution of the air flow rate in the blowers (see Figure 11, left). Such inhomogeneous air flows can practically lead to an inhomogeneous pressure distribution on the housing, which in turn can cause unwanted structural or acoustic problems [14]. The inhomogeneous velocity profile extends to the adapter connecting the blower and air-measuring section and to a metal plate with a defined orifice. Passing through the orifice of this plate, the propagation of the air jet can be considered as a quasi-free jet. Immediately after this area in the air-measuring section a throttle system is located. There are large differences in the air velocity profile here (see Figure 10, right) because in this area, depending on the case under consideration, air can pass through a number of narrow orifices. After passing through the throttle area the air freely spreads in an area of atmospheric pressure.

According to the numerical results, obtained in OP1 at Flow Rate 0 m^3/min (fully closed valves in the throttle system), it can be maintained that there is a good correspondence with the experimental results. In OP1, a deviation of less than 2% from the experimental values for both Δp_{tot} and Δp_{st} was found. For OP2 the deviation for both Δp_{tot} and Δp_{st} was even less compared to OP1 and in the range below 1%. The trend for good correspondence of the numerical results with the experimental ones is consistent in the other two OPs. The largest difference for Δp_{st} is found in OP3, but it should be noted that this difference does not exceed 3%.

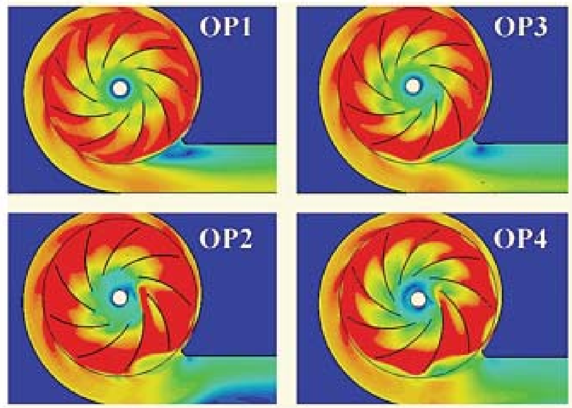

FIGURE 12. Visualization of the velocity distribution in OP1 to OP4.

The air flow behavior at different positioning of the throttle mechanism, allowing volumetric flow rate in the range from $0 m^3/min$ to $48,2\ m^3/min$ is visualized in Figure 12.

CONCLUSION

In this work, local OPs were determined using STAR-CCM+ software and v^2f Low-Reynolds Number k-ε turbulence model. The OPs allow building a numerical ventilation characteristic curve of radial high-pressure air blowers, model S-HP 470/35-105/20 in the range of flow rate from 0 m^3/min to 48,2 m^3/min. The so obtained numerical ventilation characteristic curve was validated with the experimentally ventilation characteristic curve obtained by measuring real blowers in the air-measuring section according to DIN EN ISO 5801. The accuracy of the obtained results is influenced by the size of the mesh and the number of iterations required to obtain a quasi-stationary process. Four global areas for meshing with two types of grids were selected. Local areas were used to obtain finer grid, and for some areas, such as the throttle system or the metal plate with the orifice, a mesh which is 20 and 25 % of the base mesh was used. In all OPs, an inhomogeneous distribution of the air flow velocity in the blower is observed, which is due to the geometry of the impeller itself and the fact that the impeller is not centrally located in the casing. The numerical results obtained at four OPs determined by the pressure values (Δp_{tot} and Δp_{st}) and the corresponding volumetric flow rate show excellent correspondence with the measured values. A minimal difference between the results from the numerical and experimental method for determining the ventilation characteristic curve is achieved in OP2, with a difference between Δp_{tot} and Δp_{st} obtained by both methods being less than 1 %. In the range of the entire studied interval from $0 m^3/min$ to 48,2 m^3/min, an average deviation of the numerical results from the real measurements in the air-measuring section according to DIN EN ISO 5801 for Δp_{st} average = 1,45 % and respectively for Δp_{tot} average = 1,57 % was found. The proposed numerical model of the air-measuring section according to DIN EN ISO 5801 allows high-precision analysis of the ventilation characteristic curve of radial high-pressure air blowers. The proposed numerical model of the air-measuring section according to DIN EN ISO 5801 allows analysis of the ventilation characteristic curve of radial high-pressure air blowers with high accuracy.

ACKNOWLEDGMENTS

This work was supported by the European Regional Development Fund within the OP "Science and Education for Smart Growth" 2014-2020, Project Competence Centre "Smart Mechatronic, Eco- and Energy Saving Systems and Technologies", № BG05M2OP001-1.002-0023.

The authors would like to thank the Research and Development Sector at the Technical University of Sofia for the financial support.

REFERENCES

1. R. K. Turton, "Principles of Turbomachinery", Book, Springer Netherlands, ISBN: 978-94-010-9691-1, https://doi.org/10.1007/978-94-010-9689-8, 1984.
2. R. K. Turton, "Principles of Turbomachinery", Springer Netherlands, ISBN-13: 9780412602108, 1994.
3. S. Korpela, Principles of Turbomachinery, Book, John Wiley & Sons, ISBN: 9780470536728, https://doi.org/10.1002/9781118162477.ch12, 2011.
4. S. Korpela, "Principles of Turbomachinery", John Wiley & Sons, ISBN 9781119518112 (Adobe PDF), 2019.
5. R. K. Turton, "Principles of Turbomachinery", Journal of Pressure Vessel Technology, 108(2) 247, https://doi.org/10.1115/1.3264781, 1986.
6. A. Wallis, "Axial flow fans: design and practice", Academic press, New York and London, ISBN 978-1-4832-2784-9, https://doi.org/10.1016/C2013-0-12031-4, 1961.
7. C. Teodorescu, Z. Gontean and I. Neag, "Mining Ventilation", Technical Publishing House, Bucharest, Romania, pp. 275-283, 1980.
8. T. Siwek, J. Górski anf S. Fortuna, "Numerical and Experimental Study of Centrifugal Fan Flow Structures and Their Relationship with Machine Efficiency", Polish Journal Environmental Studies, Olsztyn, Poland, Volume 23 Issue 6, pp. 2359-2364, 2014.
9. A. Sheard, , A. Corsini, S. Minotti and F. Sciulli, "The role of computational methods in the development of an aero-acoustic design methodology: Application to a family of large industrial fans", 14th International Conference on Fluid Flow Technologies, 9-12.09.2009, Budapest, Hungary, http://dx.doi.org/10.13140/2.1.4855.8089, 2009.
10. Y. Galerkin and L. Marenina, "Investigation and Perfection of Centrifugal Compressor Stages by CFD Methods", Journal of Mechanical, Mechatronic and Manufacturing Engineering, Volume 9, Issue 1, pp. 163-168, 2015.
11. S. Khelladi, C. Sarraf, F. Bakir and R. Rey, "Study of a High Rotational Speed Shrouded Centrifugal Fan : Aerodynamics and Effects of shroud associated cavity on the Performance" Journal of Power and Energy, Volume 224, Issue 5, pp. 691-700, https://doi.org/10.1243/09576509JPE899, 2010.
12. V. Stoyanov, V. Nikolov and M. Garcia, "Numerical study of heat transfer between impinging gas jets and solid surfaces", IOP Conf. Ser.: Mater. Sci. Eng., IOP Publishing Ltd, Volume 618, 012064, https://doi.org/10.1088/1757-899X/618/1/012064, 2019.
13. E. Petukhov, Y. Galerkin and A. Rekstin "A CFD-calculation method for vane diffusers of a centrifugal compressor stage", IOP Conf. Ser.: Mater. Sci. Eng., IOP Publishing Ltd, Volume 604, 012051, https://doi.org/10.1088/1757-899X/604/1/012051, 2019.
14. M. Younsia, A. Djerradab, T. Belamric and F. Menterd, "Application of the SAS turbulence model to predict the unsteady flow field behaviour in a forward centrifugal fan", International Journal of Computational Fluid Dynamics, Volume 22, Issue 9, pp. 639–648, https://doi.org/10.1080/10618560802432944, 2008.