A R C H I V E O F M E C H A N I C A L E N G I N E E R I N G

VOL. LXI 2014 Number 1

10.2478/meceng-2014-0004

Key words: centrifugal recirculating blower, semi-open impeller, numerical simulations

WŁADYSŁAW KRYŁŁOWICZ *, *MICHAŁ KUCZKOWSKI* **, *KRZYSZTOF SOBCZAK* *

DESIGN AND EXPERIMENTAL AND NUMERICAL VERIFICATIONS OF THE RECIRCULATING BLOWER FOR LONG-TERM TESTS OF TURBINE FLOWMETERS

A design of the centrifugal recirculation blower as well as results of its experimental and numerical investigations are presented in this paper. The blower was designed to work in the unique test stand which is used for long-term tests of turbine flowmeters. A 1D method was used to design this blower, then experimental and numerical studies were conducted in order to verify the 1D method. A comparison of the blower pressure increase obtained from the experiment and the computations is presented. Velocity and pressure distributions from the numerical simulations in selected sections are also shown and discussed. Additional numerical studies of a shrouded rotor and a rotor with a lower tip clearance were conducted and are presented in the paper as well.

1. Introduction

Turbine flowmeters are devices designed to measure a volume flow rate of the gas flowing through the installation. The gas flowing into meters causes rotation of the turbine rotor. The turbine rotational speed is proportional to the flow velocity, and thus, to the gas flow rate. This kind of measurement technique is classified as "traditional technologies" as opposed to non-intrusive Coriolis, magnetic, ultrasonic flowmeters classified as "new technologies" [1]. Due to high accuracy (±1%) of turbine flowmeters, they are widely used, e.g., in crude oil, refined product and gas pipelines.

The Polish standard PN-EN 12261:2002+AC:2003 "Flowmeters – turbine flowmeters" imposes an obligation of durability trials of flowmeters on

* *Instytut Maszyn Przepływowych, Politechnika Łódzka, ul. Wólczańska 219/223, 90-924 Łódź, Poland; E-mail: wladyslaw.kryllowicz@p.lodz.pl; krzysztof.sobczak@p.lodz.pl*

** *Instytut Inżynierii Środowiska i Instalacji Budowlanych, Politechnika Łódzka, al. Politechniki 6, 90-924 Łódź, Poland; E-mail: michal.kuczkowski@p.lodz.pl*

manufacturers. This standard requires the flowmeter to be tested at the maximum gas flow for 1000 hours of continuous work. A special test stand was built to fulfill this requirement. A key element of this test stand is a blower. Long time continuous operation and one piece production of the blower determined its design. Due to manufacturing costs, a semi-open impeller was chosen and its bearing system (an impeller is fixed to the motor shaft) required a relatively high tip clearance.

The effects of a tip clearance on the performance of centrifugal blowers with semi-open (unshrouded) impellers have been a subject of many investigations, e.g., [2], [3], [4]. An application of the impeller in such a configuration causes that a flow is more complex and is characterized by higher dissipation of energy. Therefore, a classical 1D design method [5] requires adjustments of some coefficients on the basis of experimental data [7], [8].

At present, Computational Fluid Dynamics (CFD) simulations are commonly used for prediction of the flow structure in turbomachinery and verification of designs. They have been applied also to investigate a tip clearance influence on the impeller performance in centrifugal fans and blowers [9], [10], [11]. They are also used in the case of the blower presented in this paper.

2. Test stand

In the case under analysis, a test stand for three month long continuous operation at the gauge pressure of 8 bar with air as a working medium was designed. It is used for trials of turbine flowmeters designed for gases such as air, nitrogen, natural gas, etc. A scheme of this test stand is presented in Fig. 1. The presented test stand is used only to determine the durability of

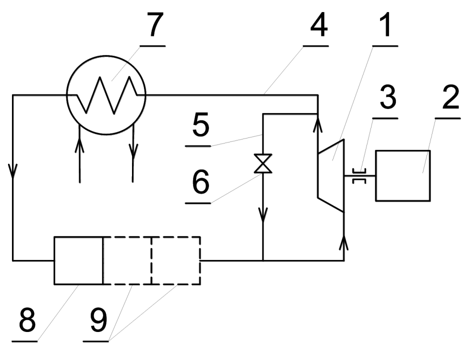

Fig. 1. Scheme of a test stand for long-term tests of turbine flowmeters: 1 – blower, 2 – electric motor, 3 – seal, 4 – pipeline, 5 – bypass, 6 – regulation valve, 7 – cooler, 8 – tested flowmeter, 9 – additional or reference flowmeters (optionally)

the flowmeter at ambient temperature. Flowmeter calibration and investigations of thermodynamic parameter (e.g., temperature) influence on the flowmeter indications are conducted on other test stands.

The blower is the ”heart” of the installation – it causes a circulation of the medium. In the case of mass production, a blower with a shrouded impeller would be considered for the test stand conditions. However, in this case, a single copy blower was required. Therefore, due to the reduction of manufacturing costs, it was decided to use a semi-open impeller, despite the fact that it is characterised by lower efficiency. It was designed at the Institute of Turbomachinery, Lodz University of Technology, to the order of the COMMON company. The blower and the complete test stand were built and they are in continuous use in the company.

3. Blower design

The primary assumption of the blower design was its one piece production. Thus, it has been assumed that all the elements can be manufactured through machining or welding. It has imposed the following design assumptions:

1. an impeller is fixed directly to the motor shaft,
2. a motor is supplied by the inverter (possibility of continuous change in rotational speed),
3. an impeller is semi-open, of a centrifugal type, machined from forged dural.

The design calculations of the blower were carried out for the parameters imposed by the manufacturer, namely:

- suction pressure p_A = 900 kPa,
- suction temperature T_A = 308 K,
- pressure ratio Π =1.0389 (static pressure increase Δp_{A-3} = 35 kPa),
- volume flow rate at the inlet (under normal conditions) $\dot{V}$ = 427 m^3/h ±4% (mass flow rate $\dot{m}$ = 1.207 kg/s).

The blower impeller is not a typical design. It is a semi-open impeller with 9 standard blades and 9 splitters. Because of the bearing system used, a tip clearance over the impeller is relatively large (0.4 mm – 3.3% of the span at the rotor outlet). Many studies, including Eckardt [6], show that the structure of the flow at the outlet of this impeller differs from the flow structure of a typical shrouded impeller (with a cover disc). Therefore, 1-dimensional calculations aimed at determination of the main blower dimensions were conducted on the basis of own experimental results. The experiments were carried out at the Institute of Turbomachinery, Lodz University of Technology [7], [8]. The results of these investigations allowed one to determine a

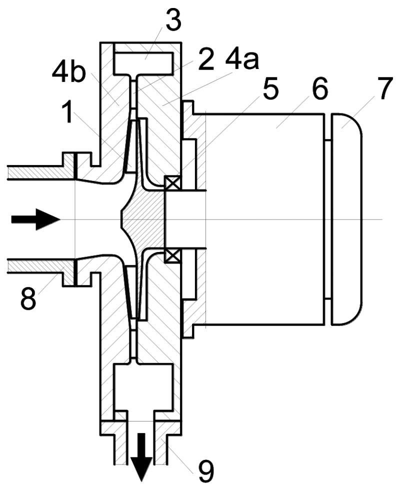

Fig. 2. Design of the blower: 1 – impeller, 2 – vaned diffuser, 3 – volute, 4a, 4b – casing, 5 – shaft sealing, 6 – electric motor, 7 – motor cooling fan, 8 – suction pipe, 9 – discharge pipe

value of the slip factor μ_u. This coefficient was then used in the calculations performed according to the classic Eckert formula [5]:

$$\mu_{\mathrm{u}} = \frac{C_{2u}}{C_{2u}*} = \frac{1}{1 + \left(\frac{h*}{h}\right) \cdot \frac{\pi \cdot \sin\beta_2 *}{2 \cdot z \cdot (1-\vartheta_n)}}$$

where: β_2* – impeller blade trailing edge angle, $\beta_2* = 58°$;
z – blade number, z = 18;
ϑ_n – diameter ratio D_1/D_2, $\vartheta_n = 0.48$;
$\left(\frac{h*}{h}\right)$ – jet-to-wake ratio at the rotor outlet. In the presented case, this value was established experimentally to be equal to 0.87.

Table 1.

Results of the 1D calculations

Location	Pressure	Temperature
	[Pa]	[K]
inlet of the suction duct	899 773	308
inlet of the impeller	899 479	307.96
outlet of the impeller	923 295	310.70
outlet of the blower	934 466	312.37

With this method it was possible to determine average thermodynamic parameters in the blower characteristic sections, i.e., at the inlet and the outlet of the impeller and at the outlet of the diffuser. The obtained values are shown in Table 1. The isentropic efficiency is $\eta_s = 0.763$. The impeller and the whole compressor stage are shown in Figs. 3 and 4, respectively.

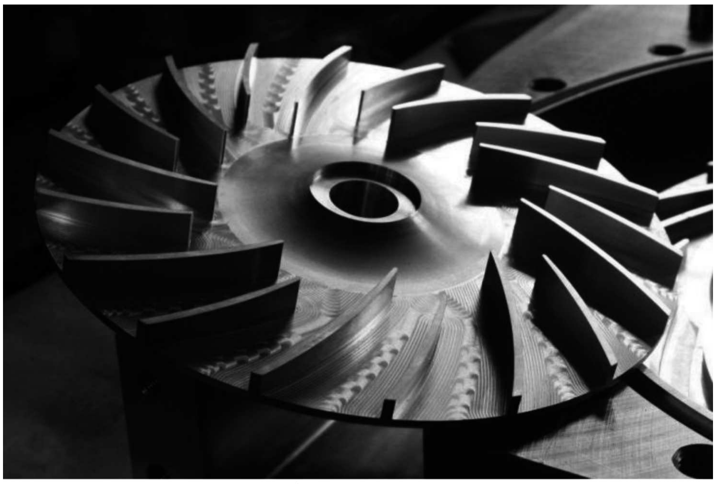

Fig. 3. Impeller

Fig. 4. Compressor stage

4. Measurements

The experimental determination of flow parameters was conducted on the blower which was installed in the test rig in order to gather flow parameters. The measurements were carried out for the nominal working pressure (900 kPa) and the nominal flow rate (1.207 kg/s).

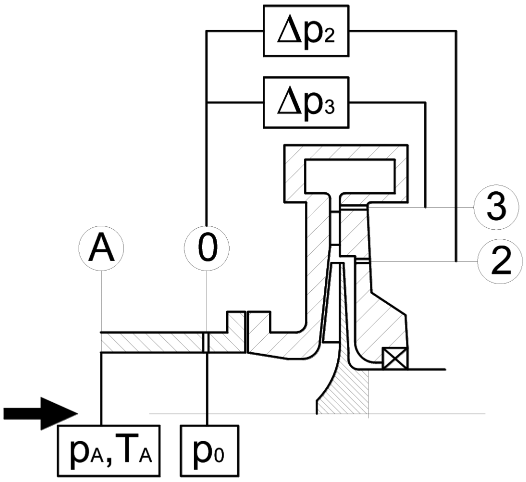

Fig. 5. Measurement scheme

A scheme of the measurement system is presented in Fig. 5. Static pressures were gathered in six taps in blower walls connected to common collectors, giving thus pneumatically averaged values in each control section. In control section A located in the suction pipe, temperature was measured by means of two thermocouples type J and pressure was measured by means of a Druck transducer. Pressure differences between sections 2-0 – Δp_2 and 3-0 – Δp_3 were measured using also a Druck transducer of uncertainty equal to ±60 Pa (in the range of 35 kPa).

5. Numerical simulations

The simulations presented in this paper were carried out for a computational model of the blower composed of an inlet duct, a rotor and a vaned diffuser. In this study, a flow through the volute was not considered. As has been mentioned earlier, the rotor consisted of 18 blades. The diffuser consisted of 19 blades.

Assuming the same flow structure in blade passages of the blower for operating points close to nominal, the calculations were conducted for 1/9 of the whole rotor wheel that comprised one impeller and one splitter blade. In the case of the diffuser, the channel was composed of two diffuser blades (2/19).

Computational meshes for the rotor and the diffuser were generated with the ANSYS TurboGrid software, which provides easy generation of high quality hexahedral meshes for blade passages in turbomachinery. As far as the solver requirements are concerned, good qualities of meshes were obtained in all cases. Approximately 1.0, 1.5 million nodes were used for the rotor and the diffuser, respectively. In the case of the inlet duct, a tetrahedral mesh generated with ANSYS Meshing was used. It consisted of approximately 0.075 million nodes.

The ANSYS CFX code was used in the simulations. Three-dimensional, compressible air flows were solved. Due to a low pressure ratio, incompressible flow simulations could be considered. However, a flow with a relatively high velocity through tip clearances over the impeller was expected and density changes could have some influence on the flow structure. Steady state simulations were conducted with the Stage interface between the rotor and the diffuser. Since its introduction in 1979 [12], this interface method has become the industrial standard in rotor-stator simulations. Between the rotating blade passage and the non-rotating vane passage, flow properties are averaged circumferentially. This kind of interface removes all transient rotor-stator interactions, but it still gives fairly acceptable results. The Shear Stress Transport turbulence model was applied with an automatic wall function. The second order space discretization was employed for governing equations.

As far as the boundary conditions are concerned, the total pressure at the inlet was adjusted to give the same value of static pressure as in the experiment and, similarly, static temperature was equal to 308 K. Additionally, because a long (more than 20 diameters) straight inlet pipe was used, a low turbulence intensity (Tu = 1%) was imposed with a turbulent to molecular viscosity ratio equal to 1. Mass flow rates from 0.6035 to 1.6898 kg/s (for the whole machine) were applied at the outlet. Channel walls were smooth and adiabatic. The rotational speed of the blower rotor was equal to 5300 rpm.

As mentioned earlier, to reduce manufacturing costs, it was decided to design a semi-open impeller. Due to the fact that the impeller is fixed to the motor rotor, a clearance between the rotor blade tip and the shroud was relatively large – 0.4 mm. The majority of simulations were performed for this impeller configuration. However, in the case of mass production, a blower with a shrouded impeller would be considered or, if a more precise bearing system would be used, the clearance of the semi-open impeller could be lower, e.g., 0.1 mm. Therefore, for these two configurations, additional simulations were carried out for the nominal point of the blower operation to learn about the efficiency reduction of the proposed design. It can give some indications for further similar designs.

6. Results

A 1D method was used to design this blower, then experimental and numerical investigations were conducted in order to verify the 1D method. A comparison of the blower pressure increase obtained from the experiment and the computations is presented for the impeller itself and the stage composed of the impeller and the diffuser. Figure 6 shows a performance characteristic curve of the impeller as a function of the relative mass flow rate (related to the nominal one). A red line refers to the data from the numerical simulations, the blue one represents the measurement results and a green triangle shows the value obtained from the 1D design procedure. The 1D procedure underpredicts the experimental pressure increase by 1.1% only. This very good correlation of the 1D design data is the effect of application of properly determined empirical factors [7], [8]. The numerical simulation data are also in a very good agreement with the experimental ones. For the nominal mass flow rate, the overprediction of the pressure increase from the numerical simulations is 0.76% only.

For the relative mass flow lower than 0.8, it is easy to notice that discrepancies are higher. It can be due to the fact that the computations were conducted for one impeller pitch (1/9 of the impeller) assuming the steady flow and the same flow structure in all passages. However, for such low values of the mass flow rate, some unsteady phenomena can take place and the flow structure is not fully identical in all passages. Therefore, the discrepancies between simplified simulations and experimental results increase when the mass flow rate decreases.

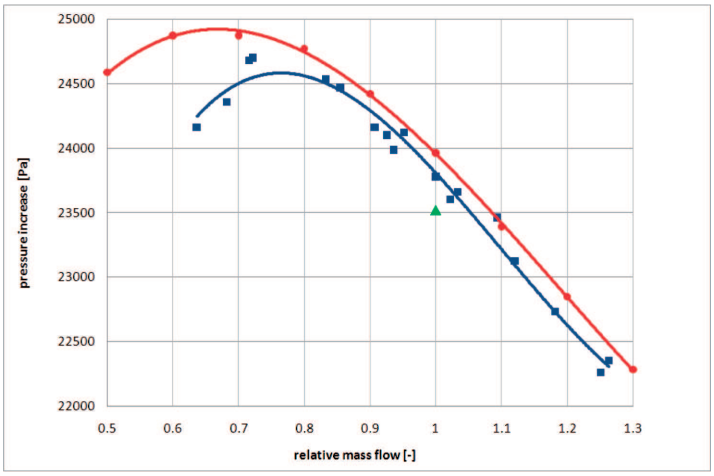

Fig. 6. Performance (pressure increase) characteristics of the impeller: red – numerical simulations, blue – measurements, green – 1D design

Figure 7 shows performance characteristics – a pressure increase between the inlet of the impeller and the outlet of the diffuser. The measurement, experimental and 1D design data are presented as in Fig. 6. The additional purple line corresponds to the isentropic efficiency obtained from the simulations. The efficiency characteristics is shown to confirm the choice of the nominal operating point.

What is worth mentioning is a perfect prediction of the 1D design pressure increase (a green triangle). In the case of the performance characteristic curve of the impeller and the diffuser together, differences between the simulations and the measurements are higher than in the case of the impeller alone. For the nominal operating point, the difference equals 4.0%. This higher difference can be due to another simplification applied during the simulation. As has been mentioned earlier, steady state simulations were preformed with the Stage interface between the impeller and the diffuser. As a result, transient interactions of the complex flow structure downstream of the impeller with diffuser blades are artificially dumped owing to the circumferential averaging of the parameters downstream of the impeller. Therefore, some underprediction of dissipative effects in the diffuser took place and the blower performance is overpredicted but still the differences in reference to the measurements are quite low. Once again, a significant growth in discrepancies between the numerical and experimental results can be observed for the mass flow rates lower than 0.9 of the nominal one, which is due to the above-mentioned simplifications in the impeller simulation method.

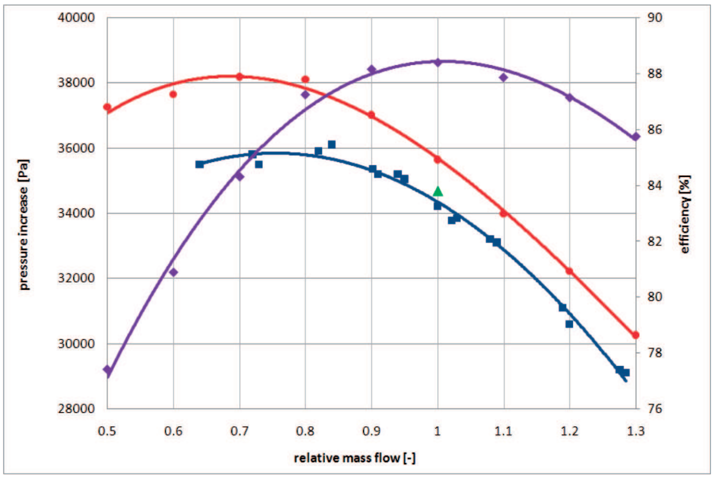

Fig. 7. Characteristics of the impeller and the diffuser: performance: red – numerical simulations, blue – measurements, green – 1D design; efficiency: purple – numerical simulations

As already mentioned, some additional simulations were performed for the shrouded impeller and for the semi-open one with a reduced tip clearance (0.1 mm). CFD simulations give a possibility of a detailed flow presentation and analysis for these designs. Figures 8 and 9 present the total pressure (whose decrease indicates regions of energy dissipation) and streamline distributions in the blade-to-blade view of the blower impeller at 10%, 50%

and 90% of the span, respectively. For the shrouded impeller, some drop of total pressure is observed close to the hub and the shroud due to secondary flows and in the middle of the channel height in the boundary layer and the wake at the blade. For the semi-open impeller, a quite well organized flow is shown at 10% of the blade height with some disturbances at the outlet

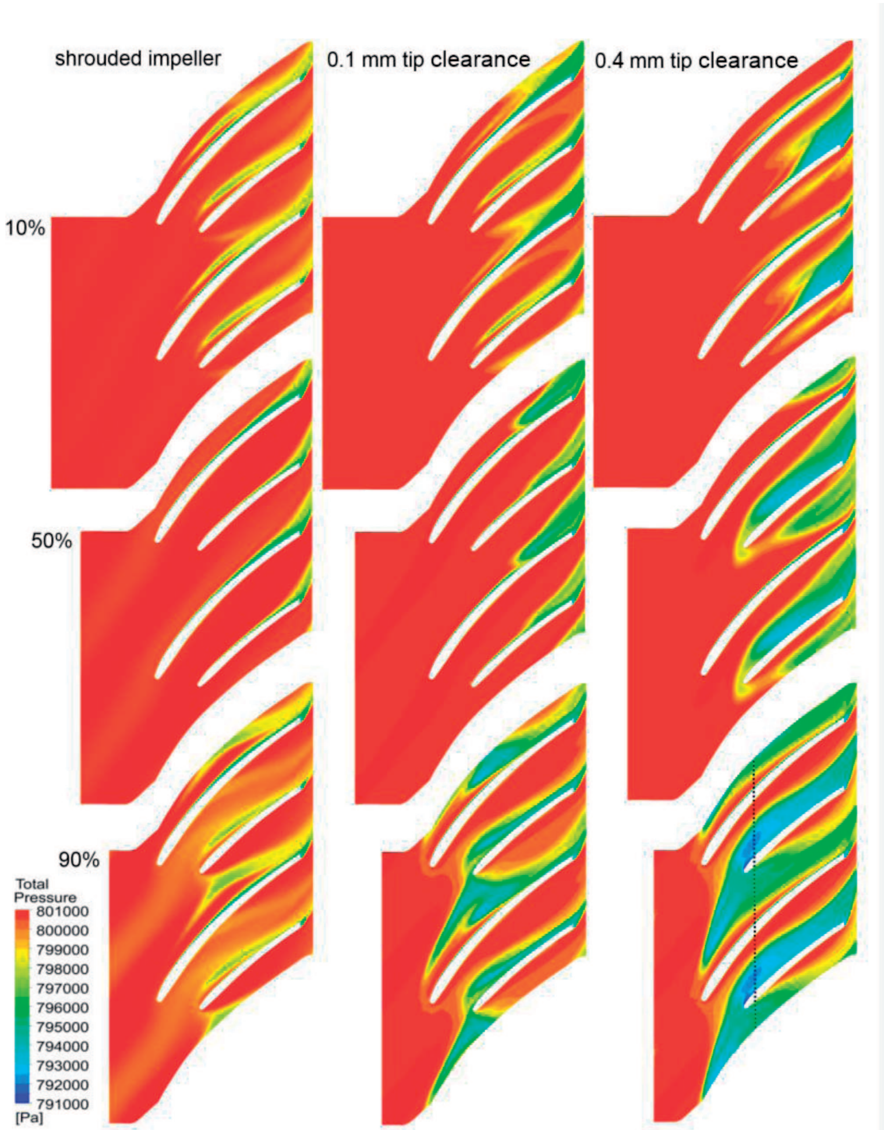

Fig. 8. Total pressure in the blade-to-blade view of the impeller at 10%, 50% and 90% of the span – left column: shrouded impeller (without a tip clearance), middle column: 0.1 mm tip clearance, right column: 0.4 mm tip clearance

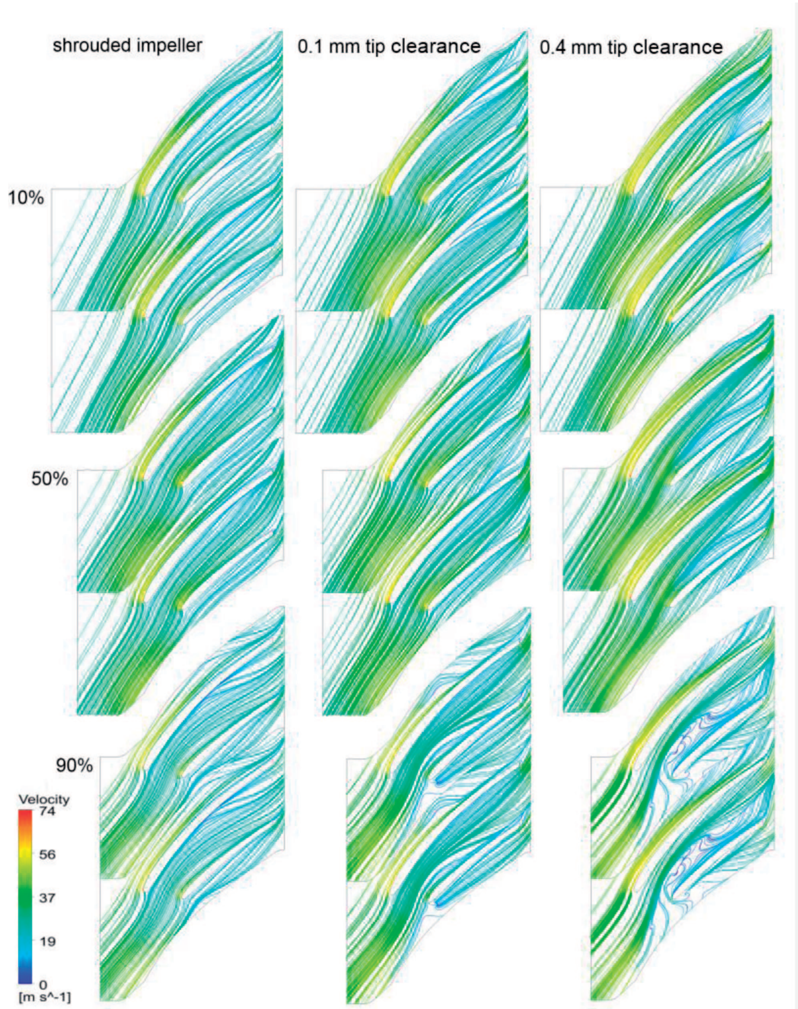

Fig. 9. Streamlines in the blade-to-blade view of the impeller at 10%, 50% and 90% of the span – left column: shrouded impeller (without a tip clearance), middle column: 0.1 mm tip clearance, right column: 0.4 mm tip clearance

region of the rotor. However, for sections above 50% of the span, a much more complex flow structure is observed. In the blower under analysis, due to the bearing system, there is a rather large (0.4 mm) tip clearance between the rotor blade tip and the shroud. Therefore, the stream of leakage is quite intensive. The leakage above the main rotor blade induces a strong vortex structure at the convex side of the blade as one can see in Fig. 10. This structure is divided by the splitter blade. In the case of 0.1 mm tip clearance, the vortex intensity is not so strong – vortex structures reach 50% of the span only in the region of the splitter trailing edge. It is easy to see a mixing zone as the region with a lower value of the total pressure and a complicated arrangement of streamlines for 90% of the blade height for both semi-open impellers, however it is much more intensive for the case with a 0.4 mm clearance. A quality of solutions of such complex flow structures can contribute to the differences in reference to the experimental data presented in Figs. 6 and 7.

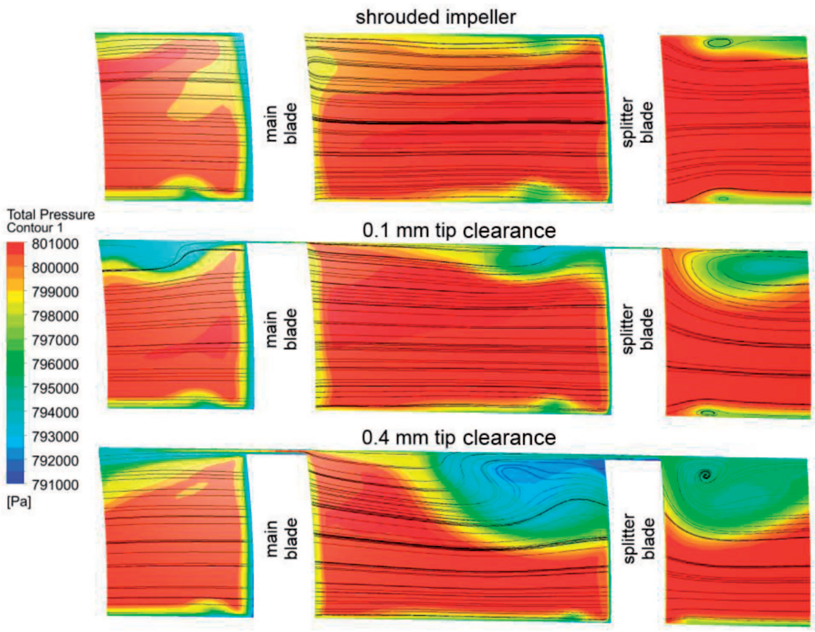

Fig. 10. Total pressure distributions on the control surface with a constant radius at 40% of the impeller length for the shrouded impeller, 0.1 mm clearance and 0.4 mm clearance

Figure 10 presents total pressure distributions and streamlines on the cylindrical surface normal to the streamwise direction at 40% of the distance between the leading and trailing edges of the impeller main blade (just behind the splitter leading edge - see a dot line in Fig. 8). This plane is not perpendicular to the blades, thus not all vortex structures are clearly visible by means of surface streamlines, but it allows one to see a size of dissipation zones where the total pressure is reduced. Secondary flows close to the hub and the shroud can be noticed for the shrouded impeller. In the case of semi-open impellers, the leakage induces vortex structures whose intensity is so strong in the case of 0.4 mm clearance that disturbances at the splitter reach 50% of the span.

A negative contribution of the flow mixing due to the tip clearance for the semi-open impellers leads to a significant reduction of the impeller and whole blower efficiency as shown in Table 2. In the case of mass production, such a decrease would not be permitted but in the case of a single copy product, the cost of manufacturing is predominant. In the case of the designed blower, it enforced also a type of the bearing system which required a significant tip clearance and additionally decreased the efficiency.

Table 2.

Comparison of the blower efficiency on the basis of the numerical simulations

	Efficiency (impeller)	Efficiency (impeller+diffuser)
	[%]	[%]
shrouded impeller	96.7	93.0
0.1 mm clearance	93.0	91.2
0.4 mm clearance	88.2	88.4

7. Conclusions

The proposed design of the blower fulfilled all the client's requirements and the machine operated continuously without problems during the predefined trial period.

Due to an untypical design of the impeller (semi-open with a quite high tip clearance), the standard 1D procedure of the blower design was supported by the empirical coefficient from own experimental investigations. As a result, a good agreement between the blower measurement data and the design parameters was reached.

Despite its relative simplicity (steady state, one pitch – 1/9th of the impeller, a stage interface between the impeller and the diffuser), numerical simulation results are in a very good agreement with the experimental data in a wide range of mass flow rates. However, below 0.8 of the nominal flow rate, overprediction of the pressure increase starts to grow, which can be due to the above-mentioned simplifications.

It can be concluded that CFD is a cost effective tool for flow prediction for the blower. It can provide some detailed flow information useful for the efficient design of the machine, supplementing the 1D procedure with data which otherwise would be determined from expensive experiments. In the case under analysis, it was possible to reveal an influence of the tip clearance leakage in the impeller on the flow structure. Due to a relatively high tip clearance, the leakage is intensive and disturbances propagate in more than a half of the rotor span. Consequently, the efficiency of the blower is significantly lower than for shrouded impellers or the semi-open ones with smaller clearances.

Manuscript received by Editorial Board, March 28, 2013;
final version, January 18, 2013.

REFERENCES

[1] Mangell A.: Flow measurement techniques, World Pumps Volume: 2008, Issue: 507, 2008, pp. 32-34.
[2] Eum H.J., Kang Y.S., and Kang S.H.: Tip clearance effect on through flow and performance of a centrifugal compressor. KSME Int. J., 2004, 18(6), pp. 979-989.
[3] GAO Li-min, XI Guang, WANG Shang-jin: Influence of Tip Clearance on the Flow Field and Aerodynamic Performance of the Centrifugal Impeller, Chinese Journal of Aaeronautics, Vol. 15 No. 3, 2002.
[4] Eum H.J., and Kang S.H.: Numerical study on tip clearance effect on performance of a centrifugal compressor, ASME, Proceedings of ASME Joint U.S.-European Fluids Engineering Conference, Montreal, USA, (2002).
[5] Eckert B., Schnell E.: Axial und Radialkompressoren. ISBN: 978-3540026464, Berlin, Springer Verlag 1961.
[6] Eckardt D.: Laser velocimeter flow studies within a high-speed centrifugal compressor impeller, Turbomachinery vol. 86, 1979.
[7] Sprawozdanie z budowy i uruchomienia stoiska 127 do badań stopni sprężarek promieniowych – część B – Badanie stopni promieniowych; Internal Report IMP PŁ No 1388, Łódź, 1997.
[8] Kryłłowicz W., Magiera R., Łagodziński J., Sobczak K., Liśkiewicz G.: Aerodynamical and Structural Design of the Diagonal Blower and its Numerical and Experimental Validation, Proc.: Vibration Engineering and Technology of Machinery (VETOMAC VIII), Gdansk, 2012, pp. 167-176.
[9] Sharmal N.Y., Karanth K.V.: Numerical Analysis of a Centrifugal Fan for Improved Performance using Splitter Vanes, Engineering and Technology, Issue 0036, 2009.
[10] Engin T.: Study of tip clearance effects in centrifugal fans with unshrouded impellers using computational fluid dynamics, Proc. IMechE Vol. 220 Part A: J. Power and Energy, 2006.
[11] Patel K.K., Patel P.M.: Performance Improvement of Centrifugal Fan by Using CFD, IJAERS/Vol. II/ Issue II/Jan.-March., 2013/01-04.
[12] Denton J., Singh U.: Time Marching Methods for Turbomachinery Flow Calculations, VKI Lecture Series 1979-7, von Karman Institute, 1979.

Projekt oraz numeryczna i eksperymentalna weryfikacja dmuchawy recyrkulacyjnej do długotrwałych testów przepływomierzy turbinowych

S t r e s z c z e n i e

W artykule przedstawiono konstrukcję odśrodkowej dmuchawy recyrkulacyjnej oraz wyniki jej badań eksperymentalnych jak i numerycznych. Zaprezentowana dmuchawa została zaprojektowana do pracy na unikalnym stanowisku badawczym, które służy do długoterminowych testów przepływomierzy turbinowych. W artykule zaprezentowano porównanie przyrostu ciśnienia uzyskane z pomiarów i z symulacji numerycznych. Pokazano również rozkłady ciśnień i prędkości dla wybranych powierzchni kontrolnych.