STUDY OF FLOW STRUCTURE THROUGH CENTRIFUGAL FAN AND ITS IMPACT ON MACHINES PERFORMANCE

Stanisław FORTUNA[1], Jan GÓRSKI[2], Tomasz SIWEK[3]
[1]PhD Eng., AGH University of Science and Technology, Poland; [2]Assoc. Prof., AGH University of Science and Technology, Poland; [3]MSc Eng., AGH University of Science and Technology, Poland

E-mail: jagorski@agh.edu.pl

Abstract:

The recognition of spatial kinematics of the flow domain and its relations to the fan impeller geometry has the key influence on the machine design and better performance parameters. In this work there have been presented some experimental results of the measurements of the absolute velocity ,its components and turbulence of the air flow through the radial fan rotor operating in the various configurations. As the main measuring-tool was used three-axial hot-wire anemometer (CTA) probe ,cooperating with prepared, author's software serving to the spatial analysis of the analyzed flow-field. What was examined particularly is the distribution of the circumferential velocity components behind the impeller rim, directly affecting the fan efficiency. As a result of the flow analysis behind the rotor rim, there was established a relationship among a degree of the turbulence, and the efficiency of the machine. On the basis of averaging in time changes of fields of the velocity behind the rim, the outflow of air from the blade passage was displayed.

Key words: fans, thermo-anemometer, kinematics of flow, turbulence, efficiency

INTRODUCTION

One of most popular topics of discussion of modern world of the politics, media, and also science is pure production and the rational energy utilization. Newly arising standards form more and more rigorous restrictions concerning efficiency of the energy conversion. In the fluid-flow machines, the factor which decides about the loss sources and magnitude and the attainable pressure rise is the geometry of the rotor and the housing and its influence on the flow structure. The improper flow-path generates a number of losses, among other things: changes in flow direction, separation, flow acceleration or deceleration, mixing, friction, etc. The detailed investigation of the spatial kinematics of the flow through the fan components and its relation to the geometry of the machine permits to create tools for the design units with better energy parameters, and consequently with a positive influence on the environment, and reduced costs of maintenance and exploitation.

The article presents some results of experimental studies over the spatial kinematics of air flow through the centrifugal fan, operating in two different configurations. The complex, three-dimensional internal flow structure is difficult for unambiguous solution on the theoretical backgrounds, therefore in the literature we usually meet the simplified models of energy conversion and kinematics in the modern centrifugal fan. The real flow is simplified to the elementary configuration, and then, using the basic conservation laws and empirical correlations concerning the efficiency and loss coefficients we determine averaged parameters of the flow-field. Analytical methods are not precise enough and often show completely unrealistic results. In the present work, the authors present experimentally developed images of the spatial flow structure at the rotor inlet and outlet of an industrial centrifugal fan.

A main measuring-tool used was the three-fiber thermo-anemometer probe, enabling measurement of the absolute velocity of air-flow, its components in the Cartesian coordinate system and their turbulence. Precise positioning of the probe and developed own software for the processing and measurement data analysis enabled generating of qualitative images of the complex flow structure. These results are useful for the design of centrifugal fan stages, optimization of its geometry, and also to the verification of theoretical models used in the computational fluid mechanics (CFD).

The authors pay a special attention on the analysis of relationships between the machine geometry and its performance. Based on the measurement of individual components of absolute velocity we documented the real influence of the flow structure on the operating parameters of the investigated stage. Circumferential velocity component at the impeller outlet rim corresponds with the increase of the fluid energy in the machine, in compliance with the rule of angular momentum preservation. Additionally the radial component of fluid velocity decides about the volume flow rate through the rotor.

During research tests we obtained detailed results of analysis of the spatial velocity distribution at the stream incoming to the rotor inlet chamber and in the exit section of the rotor operating both in the open space and covered by the volute housing, analyzed for two angles of spiral diffuser. For adopted measuring systems we carried out averaging of individual velocity components along the rotor width. Generated 3-D velocity profiles have been collected on the attached common diagrams.

The turbulence level and flow unsteadiness were also analyzed. The work consists results both for an isolated rotor and closed in the volute scroll. In both cases, exists a zone of energy dissipation associated with high turbulence levels. Evaluation of the influence of the turbulence level in the flow on the machine efficiency has been made. Averaging values of turbulence at the rotor outlet for different operating conditions were correlated with the fan efficiency curve. The approximate linear correlation between an internal work and turbulence has been confirmed for a rotor operating in the open space and inside the spiral housing.

MACHINE PERFORMANCE AND THE FLOW KINEMATICS

An elementary model used in 1-D analysis of the flow in turbo-machines is a well known description given by Euler which is based on the rule of the angular momentum conservation. It says that the sum of moments of all external forces acting on the fluid, in relation to the axis of rotation is equal to the change of its angular momentum in given time interval [3]. It is described by the formula resulting from Newton’s 2-nd law of motion

$$\sum \bar{M} = \frac{d\bar{K}}{d\tau} \tag{1}$$

The description created by Euler refers, however to a very simplified model of the flow, based on following assumptions: the fluid is incompressible and inviscid, the machine rotor consists of an infinite number of zero-thickness blades, the flow through the rotor is axially symmetrical, and there is no pre-rotation of the flow at the rotor inlet. After integration of the equation (1) at the steady uniform flow we obtain

$$M = \dot{K}_2 - \dot{K}_1 = \dot{m} \cdot c_2 \cdot l_2 - \dot{m} \cdot c_1 \cdot l_1 = \dot{m} \cdot (c_2 \cdot l_2 - c_1 \cdot l_1) \tag{1a}$$

From the trigonometric relations for the radial machine, (fig. 1) result following dependences

$$l_2 = r_2 \cdot cos\ \alpha_2, \qquad l_1 = r_1 \cdot cos\ \alpha_1. \tag{2}$$

After substitution eqns. (2) to the eq. (1a) we receive

$$M = \dot{m} \cdot (c_2 \cdot r_2 \cdot cos\ \alpha_2 - c_1 \cdot r_1 \cdot cos\ \alpha_1). \tag{1b}$$

From averaging triangles of the speed on the inlet and the outlet to the rotor results

$$c_{2u} = c_2 \cdot cos\alpha_2, \qquad c_{1u} = c_1 \cdot cos\ \alpha_1\ , \tag{3}$$

and after the substitution of the equation (3) to (1b)

$$M = \dot{m} \cdot (c_{2u} \cdot r_2 - c_{1u} \cdot r_1). \tag{1c}$$

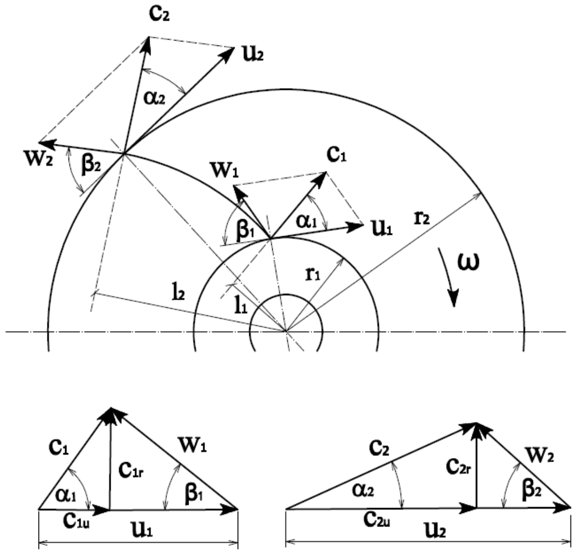

Fig 1. *Kinematics of the radial rotor cascade.*

The value of the machine theoretical power is the product of torque and angular velocity ω

$$N_{ut\infty} = M \cdot \omega = \dot{m} \cdot (c_{2u} \cdot r_2 \cdot \omega - c_{1u} \cdot r_1 \cdot \omega) = \dot{m} \cdot (c_{2u} \cdot u_2 - c_{1u} \cdot u_1), \tag{4}$$

where

$$u_2 = r_2 \cdot \omega, \qquad u_1 = r_1 \cdot \omega. \tag{5}$$

The power can also be expressed as the product of mass flow and specific work

$$N_{ut\infty} = \dot{m} \cdot l_{ut\infty}, \tag{4a}$$

from this results that the theoretical work has a direct relation to the kinematics of the flow

$$l_{ut\infty} = c_{2u} \cdot u_2 - c_{1u} \cdot u_1 \tag{6}$$

This specific work concerns the rotor with the infinite number of blades. In the real machine with the finite number of blades, this relation should be corrected by the power reduction factor μ. The formula defining the theoretical work in the real rotor expresses the simultaneously specific internal work

$$l_{ut} = \mu \cdot (c_{2u} \cdot u_2 - c_{1u} \cdot u_1), \tag{6a}$$

In many cases, when there is the lack of preliminary steering wheels, the term $c_{1u} \cdot u_1$ is often omitted in connection with the lack of the angular momentum on the inlet. However, from own experimental research [7], results that frequently exists the angular momentum on the rim of blade inlet and is generated as a result of the presence of inlet chamber walls and propagation the flow disturbances backward to the principal flow direction.

What directly decides about the machine performance is averaging the radial component of absolute velocity in relation to its geometry. For an incompressible fluid the volumetric flow at the rotor outlet can be determined by the equation

$$\dot{V}_2 = c_{2r} \cdot A_2 = c_{2r} \cdot \pi D_2 \cdot b_2 \tag{7}$$

In consideration of the complex, spatial kinematics of the velocity distribution and their components in the machine, and also the propagation of disturbances and non-stationary effects, more deep analysis concerning the relationships of flow structures with the particular machine geometry is necessary.

METHODOLOGY OF THE RESEARCH

The investigation of the spatial structure of the air flow was performed on the model-radial fan. Two main cases were considered: "F2" -when the rotor of the fan works in the free space cooperating with the suction pipeline, and "F2+ob"-when the rotor is shut-in housings. In fig. 2 there is the cross-section together with the rotor cascade (fig.2a) and the isometric view of the investigated fan (fig.2b).

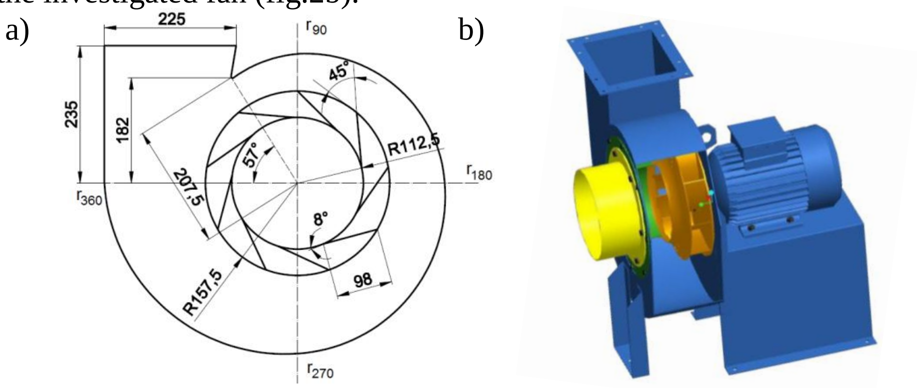

Fig.2. *The analyzed fan* F2, *a) cross-section, b) general view.*

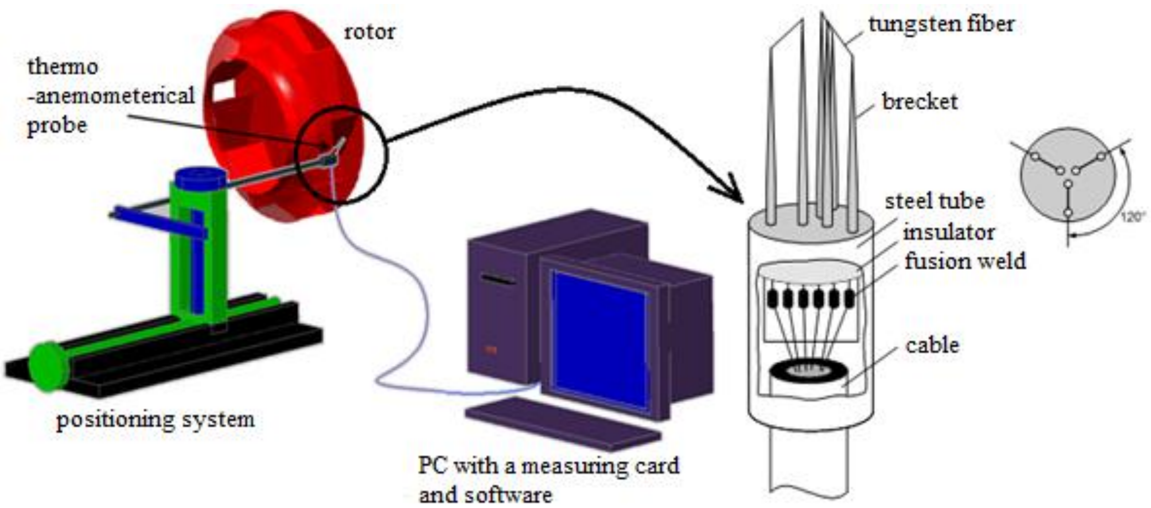

Fig.3. *Positioning of CTA thermo-anemometrical probe.*

The measuring-set to thermo-anemometer's research consisted of the joint thermo-anemometrical triple wire sensor TURBULENCE METER type ATM 94, operating with the measuring-card A/C PC LabCard PLC 814 installed in the PC computer, together with the measuring-software prepared by the Laboratory of Flow Metrology (LFM) of the Strata

Research Mechanics Institute of the Polish Academy of Sciences [1, 2]. The CTA sensors were calibrated in the LFM for their proper velocity response.

The triple wire thermo-anemometer probe is installed on the arm of positioning system equipped with two screws, whose task is the displacement of the sensor in parallel direction to the inlet suction pipe and keeping suitable height of the sensor, what illustrates the fig.4.

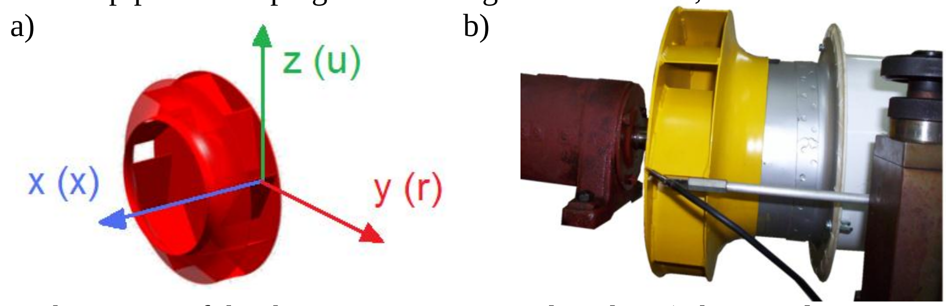

Fig.4. *The setting of the thermo-anemometrical probe, a) the coordinate system, b) the position of the CTA probe behind the rotor.*

For the purpose of the unification of velocity components measurement in characteristic sections of the investigated fan rotor, the co-ordinates system, explicitly related to the position of individual thermo-anemometer's fiber was assumed (fig.4a). The axes were chosen so that they would be compatible with directions of absolute velocity vectors: the axis „x" – for axial component c_x (blue), the axis „y" – radial component c_r (red), and the axis „z" – circumferential component c_u (green marked fiber).

DISPLAYING OF THE SPATIAL FLOW STRUCTURE

Measurement of flow-field structure was conducted in five planes and for two rotor configuration systems i.e. for the rotor operating in the housing (F2+ob) and in the free space. Sketch of the metrological basic configurations is presented in the fig. 5.

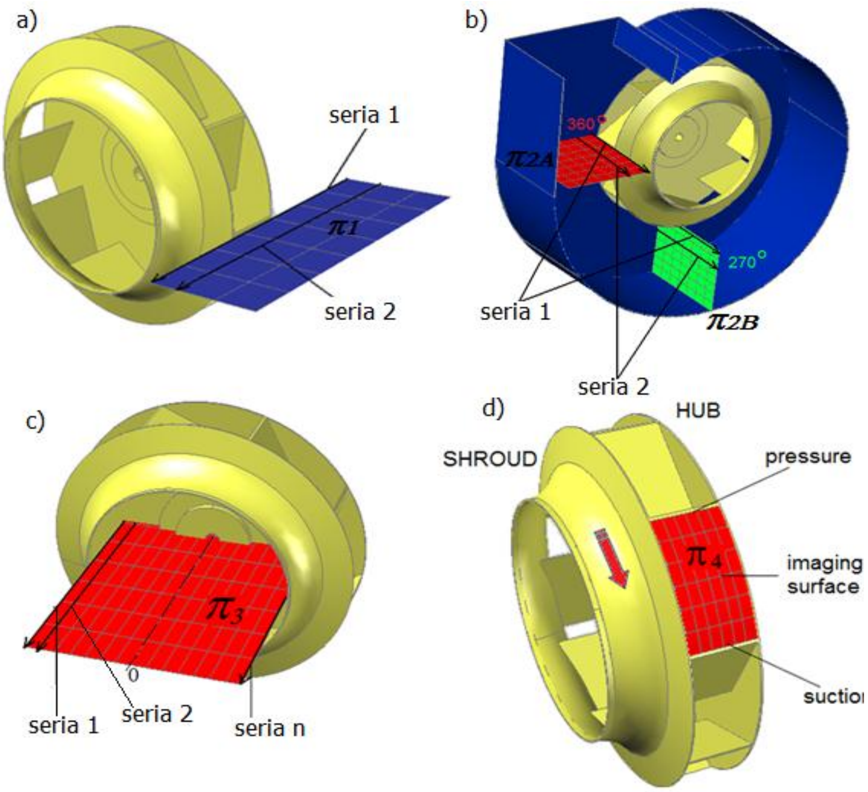

Fig.5. *Measuring-planes, a) behind the rotor in free space, b) behind the rotor in the housing, c) in the inlet chamber of the rotor, d) at the blade-channel.*

Final results of numerical calculations on measurement data are presented in graphical form of 3D diagrams - representing velocity fields and the “isotach” maps (lines of constant velocity), see figs.7c-d. In this work, by virtue of editorial restrictions, only some selected results of carried out research have been presented.

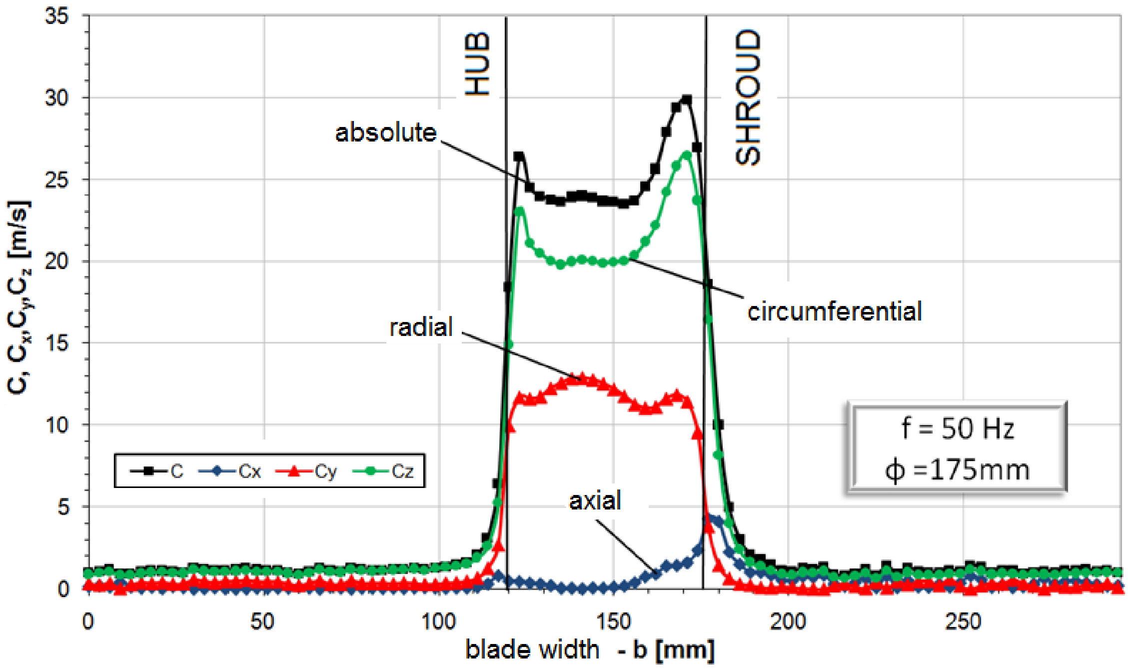

Fig.6. *Velocity distribution behind the rotor obtained in the 1-st measuring-series (case* F2*).*

Velocity distribution behind the rotor operating in the free space

The distribution of velocity behind the rotor was received as a result of setting-up, on the space and level diagrams, 17 series of thermo-anemometer's measurements made along the width of the rotor on the probing plane Π_1 from the fig.5.a. Interpolated results were presented on following diagrams. Fig.7 presents the distribution of the absolute velocity behind the rotor operating in free space. Analyzing the obtained distributions of velocity for the rotor working in free space, it is visible that the air stream flowing out from the rotor to free space has a tendency to deflect to the rear disk surface.

It is caused by an occurrence of the large component value of the axial velocity at the front disk, (fig 8.c), which "twists” the stream rearwards. A source of this component is the shape of the forepart of the rotor, where the gentle passage from the axial direction on the pass-piece takes place, and then violently on the radial direction. In the area of the passage on the radial direction appears the area of the detachment of the stream and the flow to the front disk is not parallel. It is visible in the fig. 8.b. that the peripheral speed (tangential velocity) c_u has the greatest values at the disk of the rotor. The influence on this can have the fact that the air stream flowing out in the central part of blade passage has a large radial component c_r what is visible in the fig.8.a. It causes this deviation of the circumferential component of velocity towards the radial direction in this part of the section. At disks this phenomenon is limited.

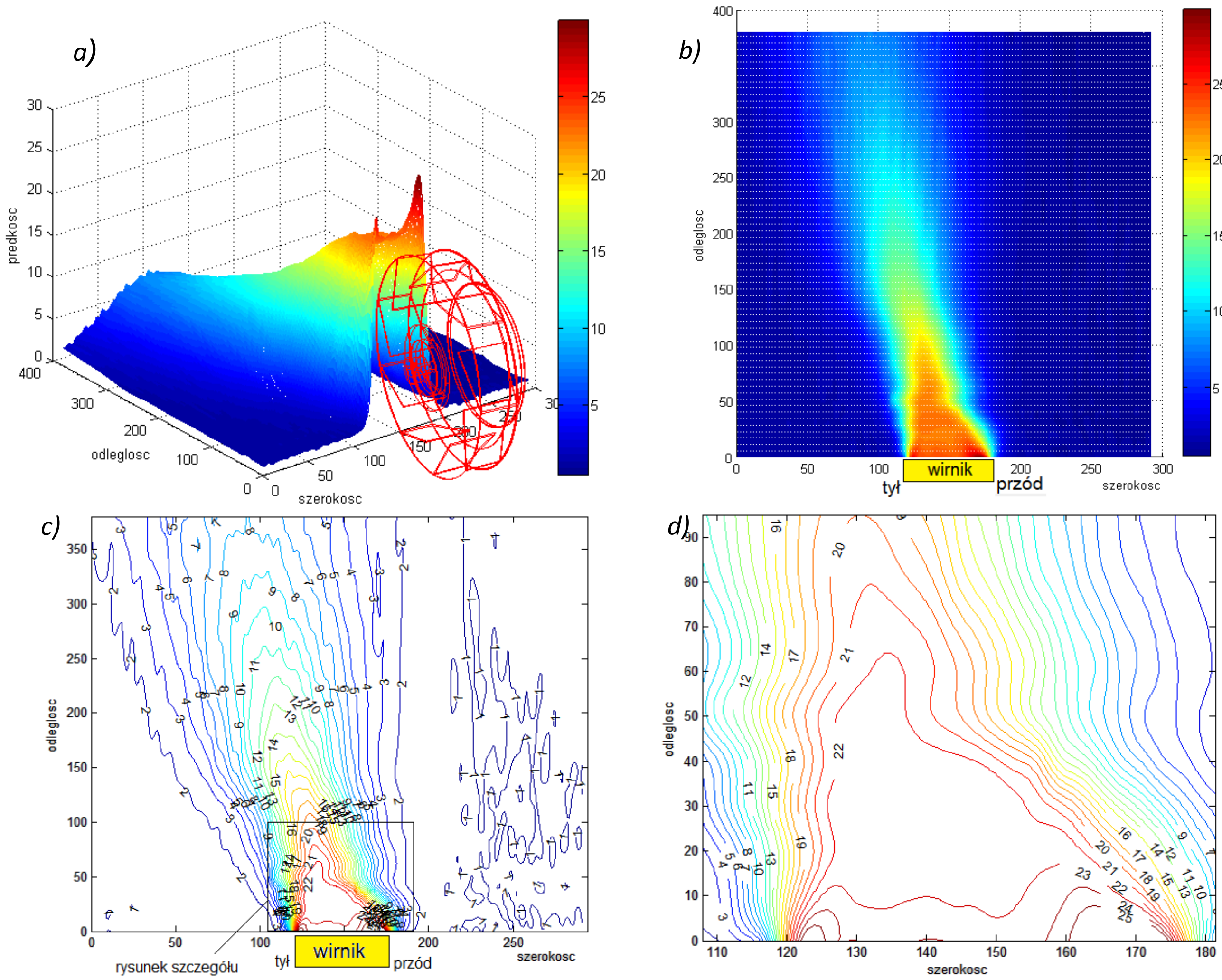

Fig.7.*The distribution of the resultant velocity c behind the rotor* F2 *working in free space, a) the space diagram, b) velocity field c) isotachs maps, d) a detail from the fig. 7.c.*

Interesting from the point of view of formation an optimum-geometry of the rotor, is recognition of the spatial structure of the flow in the blade passage. By virtue of restrictions of methods of classical thermo-anemometry, it was possible to generate the distribution of absolute velocity and its components only in exit passage profile (the plane Π_4, see, fig. 5.d). The experiment, additionally, required the necessity of forming the procedures of measuring points identification time dependence of distribution of velocities in relation to the rotor blades [8]. Applying, the already mentioned, two-dimensional interpolation by means of the „cubic" method, we obtained the 3-D images of the radial velocity components in the blade passage of the operating fan rotor, fig. 8.d. It can be noticed that the flow is non-uniform in the blade passage exit. Additional important information, which can be read on the basis of the fig. 8.d. is that the large "jet" stream passes close to the suction side of the rotor blade. The domination of the flow in the vicinity of low pressure side differs in relation to the radial compressor where the jet-flow structure is concerned near the pressure side of the rotor blade.

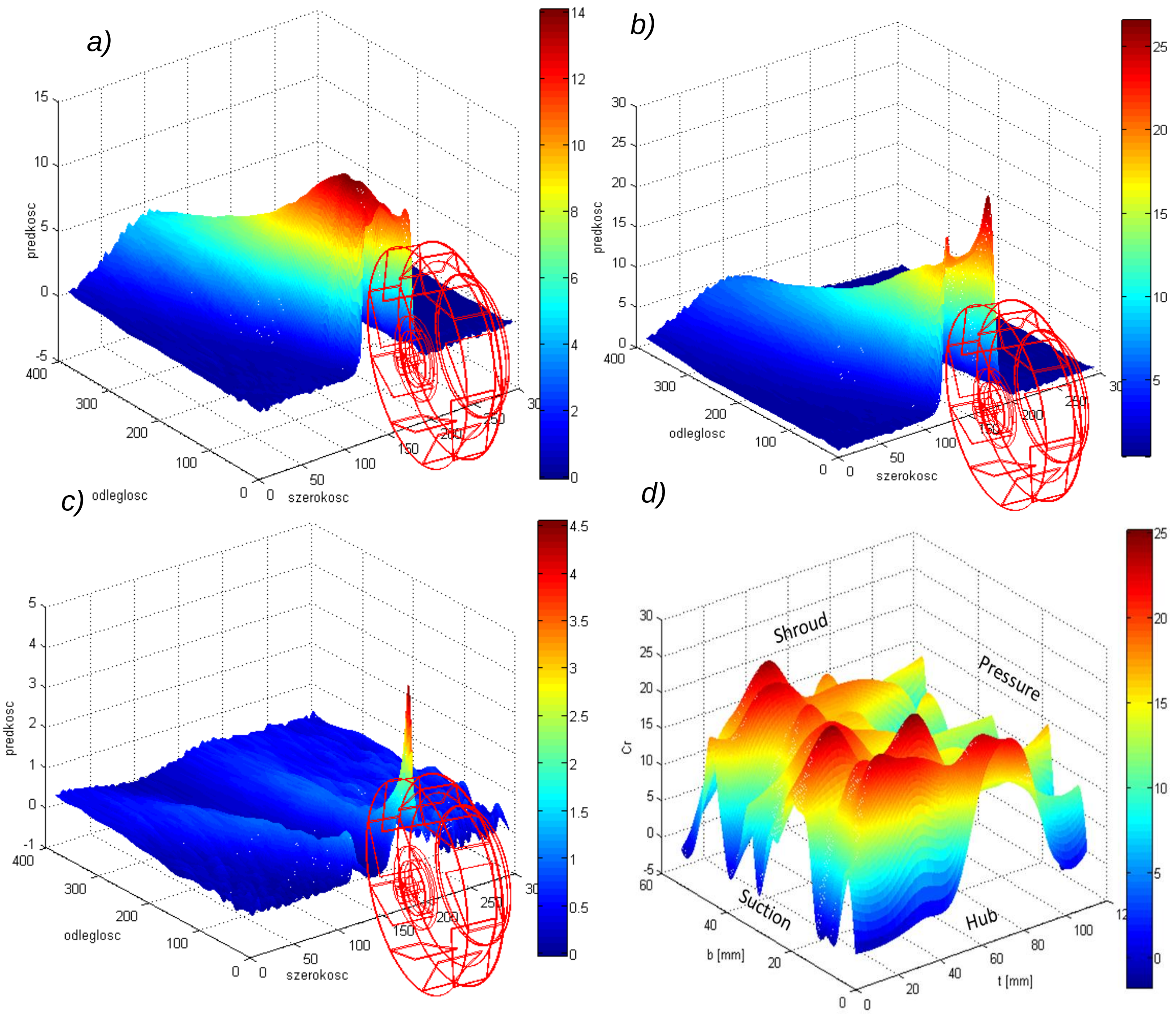

Fig.8. *The structure of the flow behind the rotor* F2, *a) radial velocity component c_{r2}, b) circumferential c_{u2}, c) axial c_{x2}, d) radial c_{r2} in the plane Π_4 - the outlet from the blade passage)*

Velocity distribution behind the rotor operating in the volute

The picture of the flow structure behind the rotor covered by the spiral housing, generated as result of penetration in the plane Π_{2a} is presented in the fig. 5.b. Measurement data were collected as a result of the displacement of the sensor probe along the rotor width from the backwall of the housing to the frontal wall. There were collected 11 velocity profiles, whereat every series included 50 measuring points, analogically to the example from the fig 6. As a consequence of the interpolation of results for, 550 measuring points, we received the distributions of absolute velocity and its components presented in the fig. 9.

Analyzing flow kinematics inside the housings we can easily notice that the distribution of the resultant velocity is nearly uniform, and is close to about 24 m/s. It is not visible here, the characteristic for these operating conditions, a high absolute velocity peak behind the exit section. The radial velocity distribution is also less important in the formation of the flow. If, for the purpose of the comparison, we accept its value equal to 15 m/s, it turns out that the area of disturbance propagation in case of the free space rotor is about 200 mms, whilst inside the housing it is only 60 mms. It is possible to assume that the exit-stream „is bent” in compliance with a direction of the impeller rotation. Both with and without the housing, the stream of flowing air is not uniform in the entire exit-section.

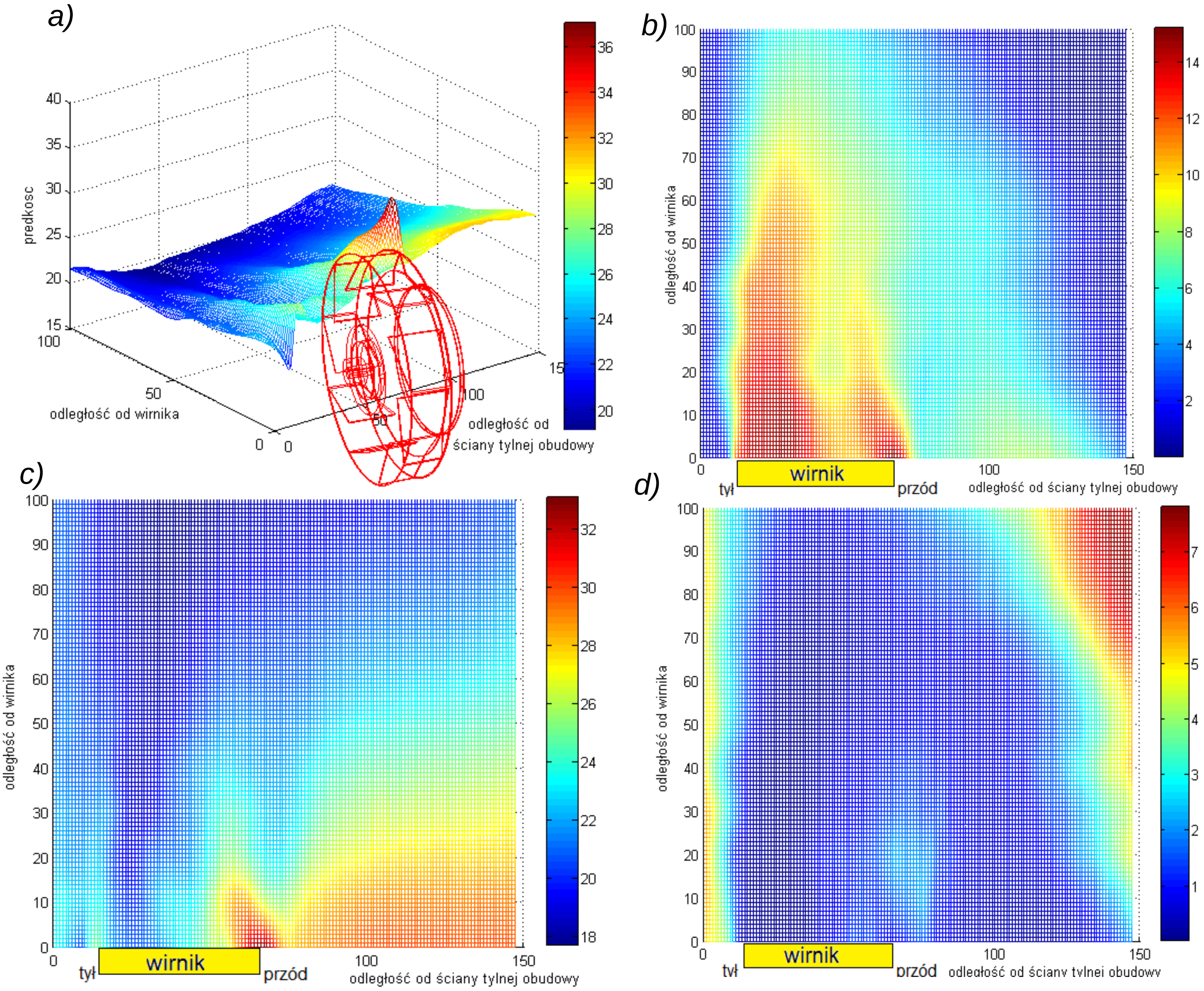

Fig.9. *The structure of the flow behind the rotor inside housings* (F2+ob)*, a) the absolute velocity c, b) constituent radial* c_{r2}*, c) circumferential* c_{u2}*, d) axial* c_{x2}

The greater outflow *I* occurs in the part close to the rear disk. In the housing we also do not observe the bend of the rearwards air stream flow. What has the influence on this is the rear housing disk- placed only 15mms behind the rotor. Around the intake-funnel forms the great whirl, in which dominates only circumferential velocity, what it is visible in fig. 9.c. Increased values of the axial component at the wall of the rear housing are a result of the occurrence, in these regions, of the distorted flow, and the boundary layer flow effects can cause the formation of small whirls. At the wall of the front housing, whirls are a result of leaks caused by presence of thermo-anemometrical probe.

Distribution of velocity inside the inlet chamber

The picture of the structure inside the inlet chamber was obtained analogically to previous experiments, sounding the plane Π_3 (fig. 5.c). The distribution of absolute velocity and its components in the inlet chamber of the rotating impeller is presented in the fig. 10.

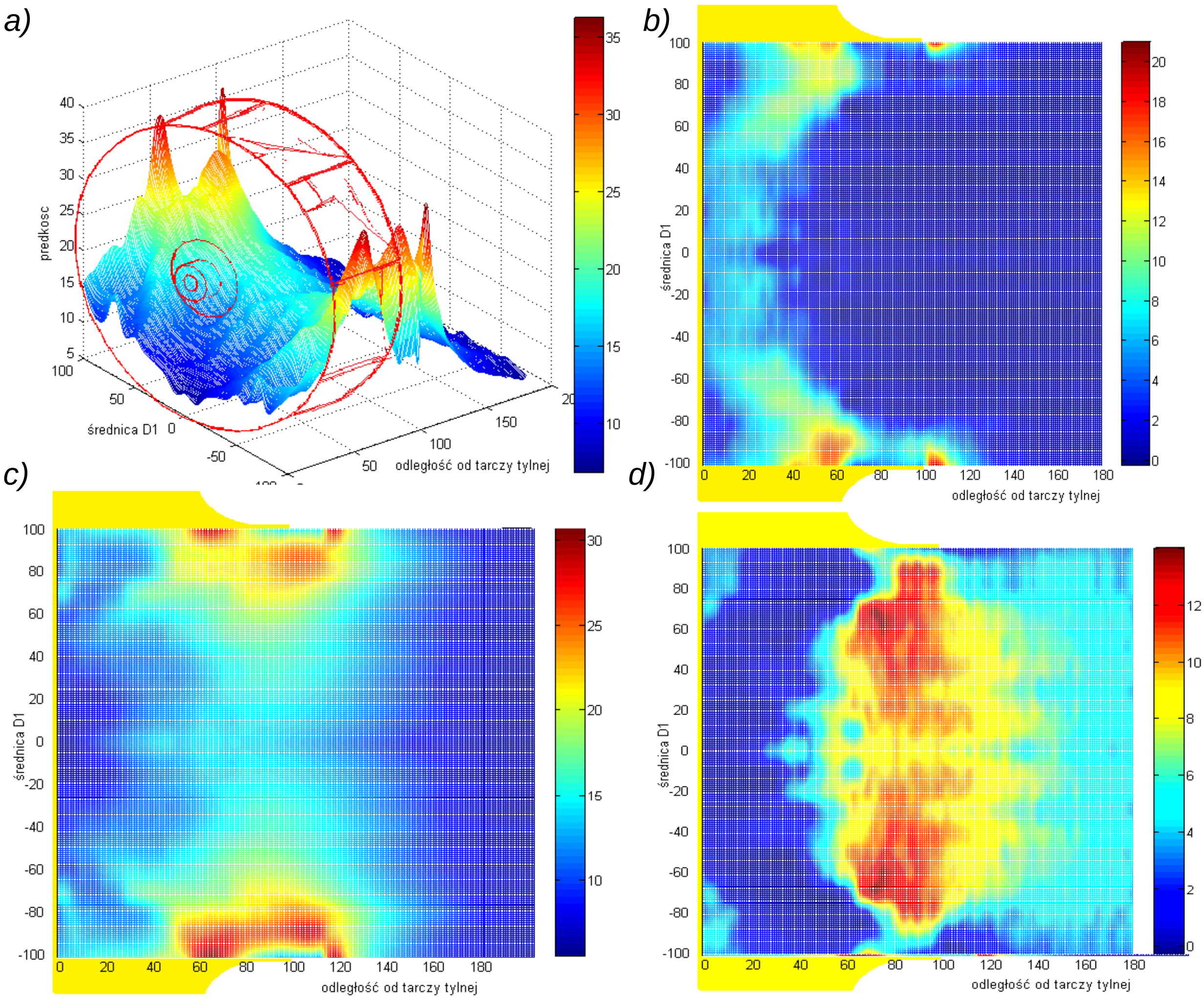

Fig.10. *The structure of the flow inside the inlet chamber, and the absolute velocity c, b) radial* c_{r2}*, c) circumferential* c_{u2}*, d) axial* c_{x2} *velocity component.*

On the basis of carried out thermo-anemometer's measurements inside the inlet chamber of the fan, we can formulate two fundamental conclusions: First the flow inside the rotor is completely axially symmetrical (investigations concern the rotor in free space). Secondly, the air flow structure is heterogeneous in the entire volume of the inlet chamber of the rotor - there are “dead zones” where air practically doesn’t flows, see fig. 11. From the distribution of the turbulence it is clearly visible that in regions of the rear disk of the rotor, between the hub and the blade rim, strongly disturbed flow zones exist [5]. In turn, from the radial velocity measurement, presented in the fig. 10.b., appears that, in this area the fluid-flow doesn’t practically shift along the diameter of the rotor.

As results from fig. 10.d., the air stream flowing to the inlet chamber of the rotor has the dominating and large axial velocity. Maximum values of this component appear in the flow through the intake-funnel. Here, in the intake-funnel appears the circumferential velocity component, which arises together with the growth of the radius r_1, and at walls of the chamber in a transition of the passage from the cylindrical part to the part of pass-piece. The radial velocity, both in profile of exit as in intake to the rim, is heterogeneous and changes along the blade width b. The influx at the intake on blade rim of the rotor is accumulated in part close to the front disk.

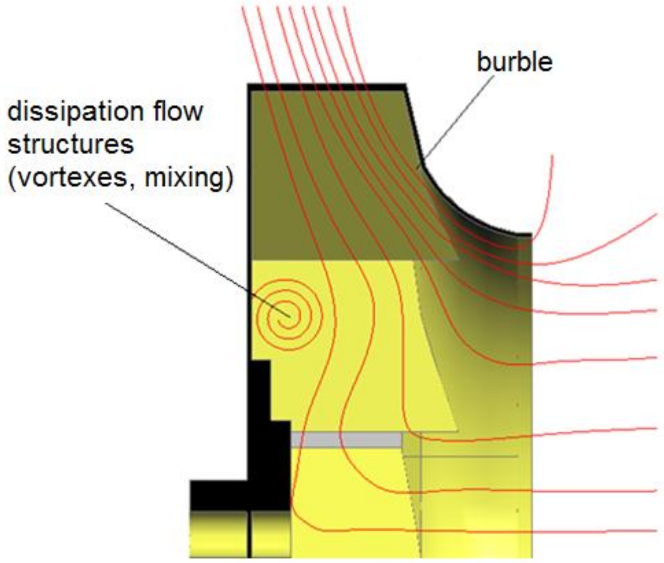

Fig. 11. The flow through the fan rotor – a simplified interpretation of flow disturbances in the meridional plane.

INFLUENCE OF THE FLOW KINEMATICS ON THE FAN EFFICIENCY

For the evaluation of the influence of absolute velocity components on performance of the machine we carried out averaging its value along the rotor width. This way, we generated suitable profiles of all velocity components. The part of aggregate results, obtained on the basis of laboratory experiments, is presented in the fig. 12.

Individual profiles are consistent with the configuration of the rotor operation from the figs. 5.a. and 5.b. Averaged series were conducted in the distance of 5 mms from the trailing edge of the rotor blade.

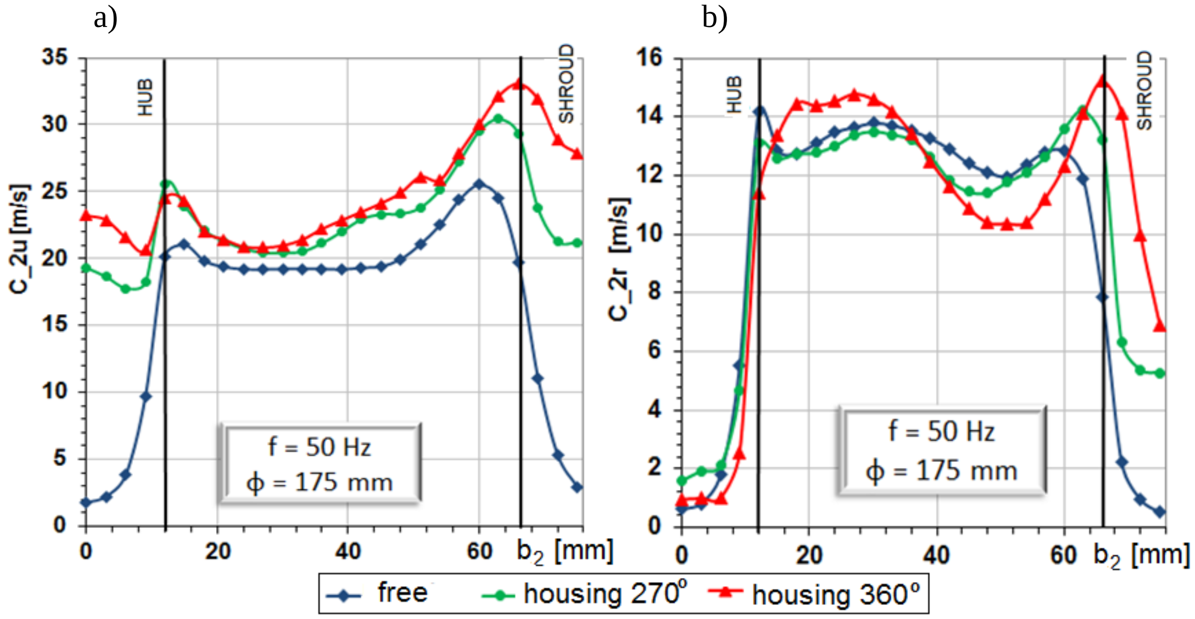

***Fig.12.** Profiles of the velocity behind the rotor a) circumferential, b) radial component*

For the purpose of the facilitation of the interpretation of results, in the table 1, average values of absolute velocity, its components and turbulences behind the rotor were collected. Averaged became only values in the rotor outlet, that is along the trailing edge of the exit blade passage.

Analyzing the distribution of the absolute velocity behind the rotor - fig.12, it is visible that this velocity for the rotor covered in the housing is about 10-12 % greater than the absolute velocity of the fluid flowing out to the free space. This doesn't mean however, that the rotor working in the spiral housing exhibits higher efficiencies.

In profile presented in fig.12.b. and the position 2 in the table 1, where we compared the radial component responsible for the efficiency of the fan, explicitly, it is visible that the configuration of the work does not have influence on its average value, and only its distribution along the trailing edge.

Axial velocity in the sum squares of individual vectors is small, so the peripheral velocity is responsible for the difference of performance of the machine, collected in fig. 8, and the position 3 in the table 1.

Table.1. *Averaging values of the speed along the blade span of the rotor b_2*

No.	**Flow parameter**	**Symbol**	**Units**	**Measurement conditions**		
				No housing (F2)	**Housing 270°**	**Housing 360°**
1.	Absolute velocity	c	[m/s]	24,7	27,5	28,4
2.	Radial component	c_{2r}	[m/s]	12,7	12,8	12,8
3.	Circumf. component	c_{2u}	[m/s]	20,6	23,8	24,7
4.	Axial component	c_{2x}	[m/s]	0,47	0,52	0,62
5.	Turbulence	T	[-]	0,31	0,31	0,33

The housing positively influences the circumferential component, significantly increasing its value. In compliance with the Euler's equation (6.a), this causes the growth of the energy transferred to the fluid, and consequently the efficiency of the machine. Interpreting the distribution of the turbulence presented in fig. 13, we can assume that as far as its value for all of three measuring-situations is equal in the plane of the outlet from the rotor and amounts about 0,31 [-], then outside disks this problem significantly becomes complicated.

a) b)

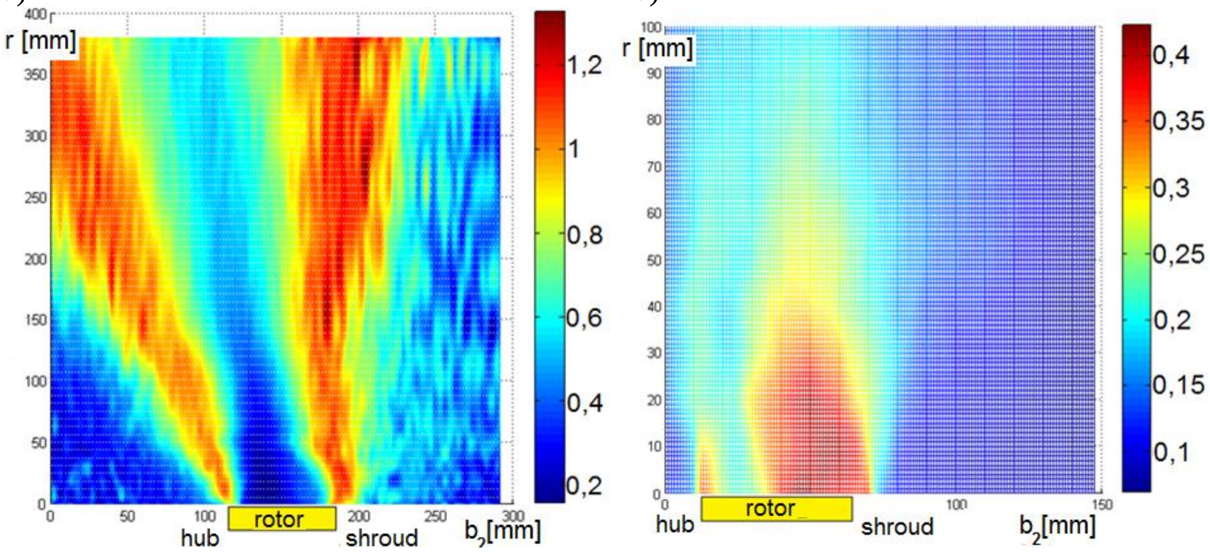

Fig.13. *Distribution of turbulence T behind the operating fan impeller,*
a) in free space (F2), *b) in the housing* (F2+ob)

In case of the lack of the rotor housing, the turbulence violently grows, what indubitable is switched with the dissipation of energy and the deterioration of the energy efficiency of the machine. In the fig. 13 we showed distribution of the turbulence in the space outside the rotor, which are also the result of thermo-anemometrical research. In the fig. 13.a. it is visible how the stream of air flowing out of the rotor interacts with the surrounding air. The situation of the rotor closed inside the housings is completely different: turbulences are for the order of magnitude smaller, and an area of its the occurrence is insignificant. The housing significantly beneficially influences the energy conversion in the low-pressure radial machine, through the improvement of the circumferential component at the rotor outlet and to the restriction of the turbulence of the flow outside the rotor.

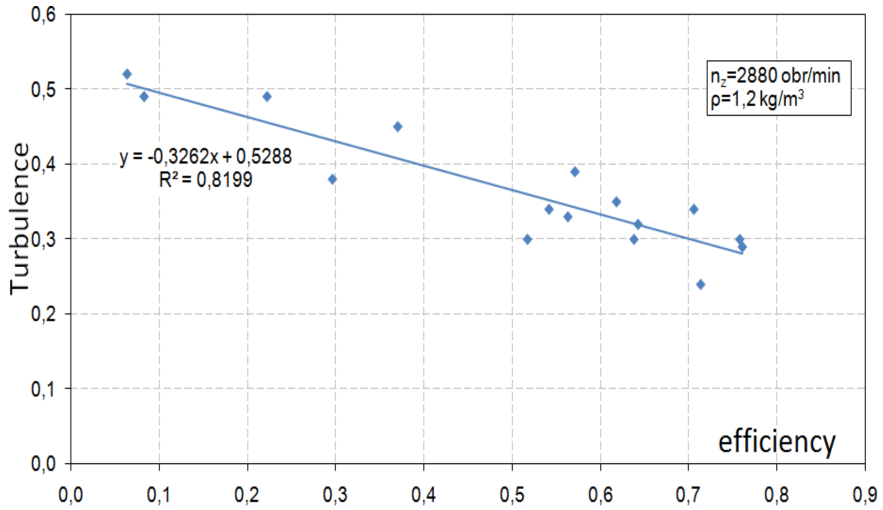

Fig.14. *Correlation between fan efficiency and turbulence intensity.*

Summary results of the analysis are presented in the fig. 14 in the form of diagram combining the relation between the turbulence intensity of the flow and the radial fan efficiency. It is easy to notice that the correlation between the efficiency of the machine and the turbulence of the flowing air can be treated as the linear function. One ought to mark that the level of the turbulence was determined according to the relation (8)

$$T = \frac{\sqrt{\frac{1}{N-1}\sum(c_i - \bar{c}\ )^2}}{\bar{c}} \tag{8}$$

CONCLUSIONS

Radial fans are one of most often met flow machines, so it is justified to conduct the continuation of research related to the settlement of functions binding the geometric shape with performance of the machine. The improvement of the efficiency of these machines will bring not only measurable economic results, but also, seemingly invisible, environmental effects, including noise reduction. Less energy consumption and improvement of the efficiency and better characteristics of exploitive fans are the measurable effects expected by the industry.

In this article, there have been presented findings on the flow structure through radial fans conducted on AGH University of Science and Technology. The group of authors works currently over the development of new constructions of radial fan rotors, seeking generalized relationships of the geometry with the proper machine performance. The collected material also constitutes the information base for testing of solutions and numerical models of the flow through the rotor, and particularly the validation of CFD techniques. The development of the reliable model, will allow to limit costs of performed experiment and its duration time.

Nomenclature

A	- area [m^2]	$\dot{m}$	- mass flow [kg/s]
b	- blade width [m]	N	- power [W], number of samples [-]
c	- absolute velocity [m/s]	r	- radius [m]
c_r	- radial velocity [m/s]	T	- turbulence intensity [-]
c_u	- circumferential velocity [m/s]	$\dot{V}$	- volume flow [m^3/s]
c_x	- axial velocity [m/s]	α	- absolute velocity angle [°]
D	- diameter [m]	μ	- power reduction factor [-]
l	- distance [m], unit work [J/kg]	ω	- angular velocity [rad/s]
$l_{ut\infty}$	- theoretical effective work [J/kg]		**Indices**
M	- torque [Nm]	$_{1,2}$	- inlet and outlet from the rotor
		$\bar{}$	- average (superscript)

References

[1] Gawor, M.; Ligęza, P.; Rachalski, A.: *Termoanemometryczne systemy wyznaczania wektora przepływu prędkości gazu*. III Konferencja Naukowa „Czujniki opto-elektroniczne i elektroniczne” COE ’94, Zegrze 1994.

[2] Ligęza, P.: *Układy termoanemometryczne – struktura, modelowanie, przyrządy i systemy pomiarowe*. Wydawnictwa AGH, Kraków 2001.

[3] Dixon, S.L.; Hall, C.A.: *Fluid Mechanics and Thermodynamics of Turbomachinery, Sixth edition*, Oxford 2010.

[4] Brzózka, J.; Dorobczyński, L.: *Matlab środowisko obliczeń naukowo-technicznych*. Wydawnictwo Mikom, Warszawa 2005.

[5] Fortuna, S.; Siwek, T.: *Imaging flow through the radial cascade flat profiles on the basis of termoanemometer’s experiment*. Problems of Mechanical Engineering and Robotics, No. 38 (1), pp. 49-62, Cracow 2009, ISBN 83-89772-37-X.

[6] Witkowski, A.; Majkut, M.: *Experimental investigation of inlet guide vane-rotor interaction in a low speed axial flow compressor stage.* Archive of Mechanical Engineering, 2006, Vol. LIII, No. 1, pp. 23-48.

[7] Fortuna, S.: *The investigation of the useful work and losses in radial fans*. (in Polish), Monographs, AGH University of Science and Technology Press, Krakow 2011.

[8] Fortuna, S.; Siwek, T.; Figiel, P.: *Visualization of fan outflow in the blade passage on the basis of thermoanemometrics measurements.* Mechanics vol. 29 No. 1, pp.11–15, Krakow 2010.