Mechanics and Mechanical Engineering
Vol. 22, No. 2 (2018) 447–454

https://doi.org/10.2478/mme-2018-0036

Numerical and Experimental Investigation of the Fan with Cycloidal Rotor

Sławomir Dykas
Mirosław Majkut
Krystian Smołka
Michał Strozik
Tadeusz Chmielniak
Tomasz Staśko
Institute of Power Engineering and Turbomachinery
Silesian University of Technology
Gliwice, Poland
krystian.smołka@polsl.pl

Received (23 June 2018)
Revised (19 August 2018)
Accepted (28 August 2018)

In this paper, the investigation of the cycloidal rotor fan (CRF) was presented. A CRF with four blades of the NACA0012 profile was used for the analysis. The CFD calculations were carried out by means of Ansys CFX commercial software. The experimental tests were done using velocity field measurement with the LDA technique. Numerical results were compared with experimental measurement in terms of velocity values. The CRF performance characteristic was prepared on the basis of experimental and numerical results.

Keywords: cycloidal rotor fan, CFD calculations, experiment, cross-flow fan.

1. Introduction

Cycloidal rotor is now considered as a new idea for aircraft propulsion systems [1, 2] and there is still not many research in this field. In recent years, interest has been growing in a new class of fans with an open rotor (without diffuser). The most popular industrial fan with open rotor are the axial propeller fan (APF). Most commonly they are used in the ventilation systems as a blowing or exhaust fan. The proposed fan with open cycloidal rotor belongs to the type of cross-flow fans (CFF), which are used commonly in the HVAC industry, as well as in electronics [4]. The cycloidal rotor fan (CRF), contrary to typical cross-flow fan, has no diffuser and its rotor allows blowing in any specified direction. It is possible by changing the incident angle, α, of each blade individually during the rotor rotation with constant angular velocity, ω (Fig. 1). The change of the incident angle during rotation from the positive value to the negative one makes possible to work each blade both as a puller or pusher, depending on the position angle, γ. In proposed CRF's design the number of the blades can be from 2 two 8 and it is dependent on the chord of the rotor blade, c, and rotor radius, R. In the presented in this paper design, the blade chord was 50 mm and the radius was 70 mm. The values of blade chord and rotor radius forced to use maximum four blades.

The volume flow rate for the cycloidal rotor fan (CRF) is affected by rotor swept area (i.e. rotor span and radius) and air flow velocity, which depends on the angular velocity (ω) and the maximum incident angle (α_0).

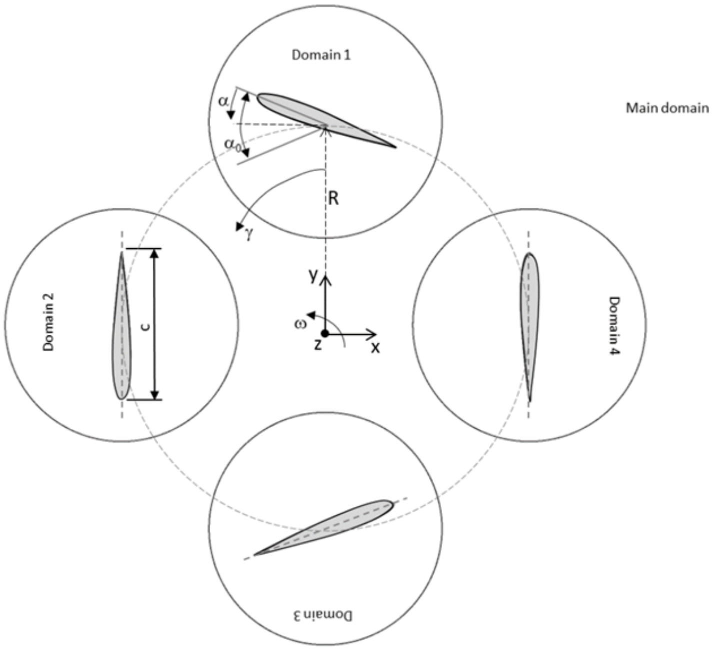


Figure 1 Scheme of the cycloidal rotor fan with four blades of the NACA0012 profiles

Based on the CFD model and the experimental results, the performance of CRF with four blades of NACA0012 profile for different rotational speed was estimated and presented.

2. Numerical and experimental models

Numerical calculations of unsteady flow field in CRF was performed by means of Ansys CFX software. To this end the URANS solver with SST k-ω? turbulence model was employed. The computational mesh consisted of in 0.6 M nodes in total, for five domains, four domains for rotors and one main domain (Fig. 2), y+ parameter in the vicinity of the wall was around 2. This number of grid points ensures the grid independent solution for considered herein test case. The grid was three dimensional, but only three layers in third (Z) direction were used. Therefore, obtained results of the flow field should be treated as a two dimensional only. The "Opening" boundary condition was set in the main domain on the outer surface. Whereas, for the connections between rotors domains and main domain the "General connection" boundary condition was used. For swinging the rotor domain while rotating CRF a special script was written in CEL, because in the standard available settings in CFX it was not possible to rotate rotor simultaneously around two different centres of rotation (for a rotor blade and entire CRF).

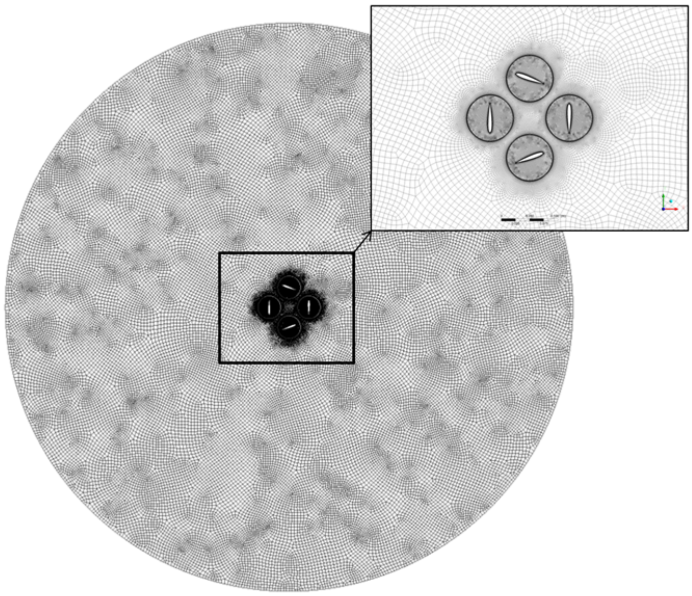


Figure 2 Scheme of the cycloidal rotor fan with four blades of the NACA0012 profiles

The transient calculations were performed with constant time step calculating on the basis of the angular velocity:

$$\Delta t = \frac{2\pi}{\omega}/n \tag{1}$$

where n is the constant used for keeping the similar time step value for numerical simulations with various angular velocity, ω.

An increment of the blade swing angle in a certain time step was calculated according to the relation:

$$\Delta\alpha = \Delta t\omega \frac{2\alpha_0}{2\pi} \frac{\sin(\omega t + \gamma)}{|\sin(\omega t + \gamma)\ |} \tag{2}$$

It can be seen from relation (2) that $\Delta\alpha$ may have a positive or negative value what ensures the return of the blade to the initial position after full rotor rotation (2π). The prototype of CRF was made by means of additive manufacturing method by the 3D printing of all elements from the polymer material (Fig. 3). The ball bearings were used for all movable connections what ensured proper behaviour of the entire design even under high rotational speed. The experimental study was focused mainly on the flow field measurements. For measurements of the velocities the LDA technique was employed.

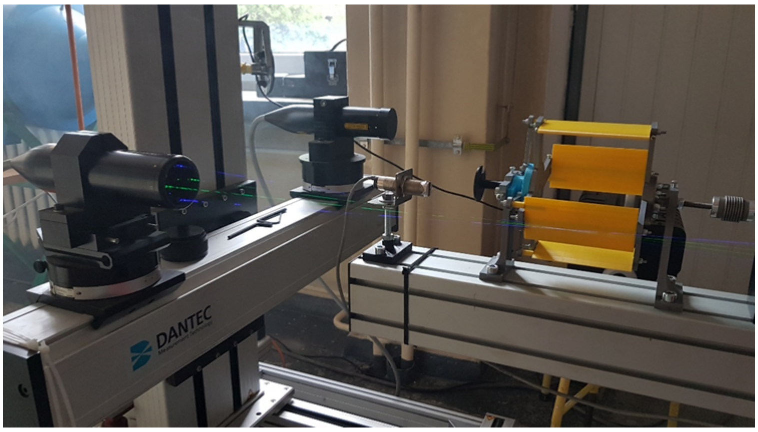


Figure 3 Photograph of the cycloidal rotor fan during the measurement campaign

The applied LDA system is a two-colour, six-beam, three-dimensional measuring system. It consists of a 4 W argon-ion laser, two 60 mm FiberFlow probes with beam translator and beam expander. Measurement of all three velocity components was done with a two-component (4-beam) probe and a one-component (2-beam) probe aligned to measure in the same position. The focal length of LDA probes is 160 mm or 400 mm with additional lenses producing a beam intersection diameter 78 nm or 194 nm and an intersection lengths of about 0.66 mm or 4.09 mm respectively. The probes are mounted on supports attached to a mechanical traverse. The traverse can move axially, horizontally and vertically, and can be rotated in horizontal plane to achieve the desired location with accuracy ± 0.125 mm. The seed particles of silicon oil with a mean diameter of $0.6 \div 1.0$ micrometre were used for improving the data acquisition process [5].

3. CRF velocity flow field

The velocity field in the cycloidal rotor fan (CRF) was determined by means of prepared numerical model in Ansys CFX as well as using the LDA technique for laboratory measurements. In Fig. 4 the velocity fields for two rotational speeds and two values of maximum incident angle are presented. It can be seen very clearly how the range of the incident angle change affects the velocity field in CRF. At higher α_0 the more significant increase of the flow velocity is observed, in spite of higher flow losses around the blades working under high value of incident angle. These losses are closely associated with flow separations on the blade surface, both on the suction and pressure sides, depending on the instantaneous incident angle value, α.

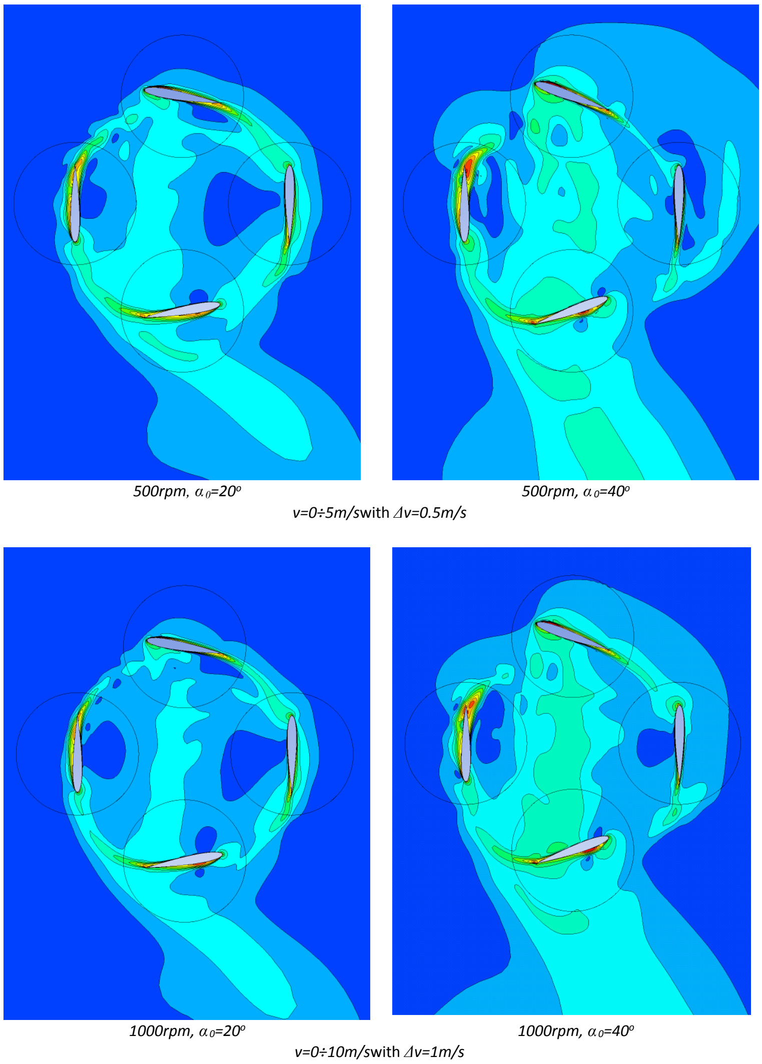
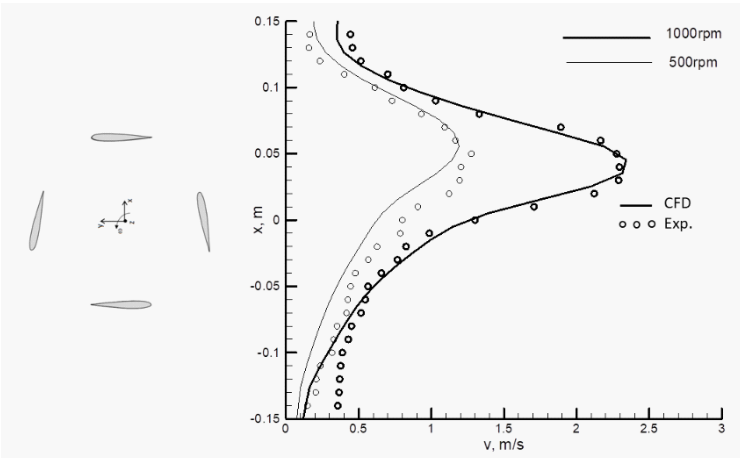


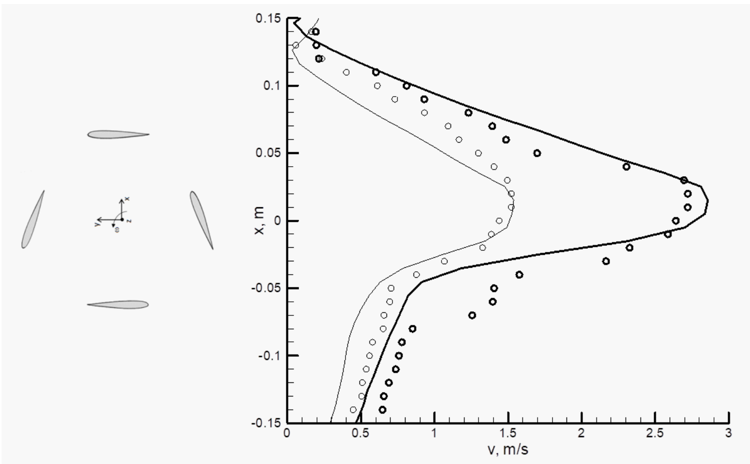
Figure 4 Velocity contour lines

On the basis of CFD calculations, one can estimate the places of flow losses generation. On this basis, the solutions for better CRF performance can be proposed. The maximum losses, manifested by area of flow separation on the blade surface, are observed for the high values of blade profile incident angle. It suggests that other blade profiles have to be considered in the future tests.

Figure 5 shows the velocity profiles in distance of $2R$ downstream of CRF centre of rotation. The velocity profiles plotted from LDA measurements are determined on the basis of velocity measurement with the pitch $\Delta x = 10$ mm. The agreement between CFD results and experimental data is satisfactory with respect to the maximum velocity value as well as velocity profiles. Visible differences may come from the more 3D effects in the experimental measurements than in the CFD model, The side walls in the CFD model were not included in the computational domain.



α_0=20°



α_0=40°

Figure 5 Velocity profiles behind the cycloidal rotor

4. CRF performance characteristic

For the fans with a diffuser, the performance characteristic is determined by flow throttling on the discharge side. Whereas, the characteristic of a fan with an open rotor is determined in a manner similar to that of a propeller. In this case, the dynamic pressure (p_d) change is affected by the change of the rotational speed and maximum value of the swing angle. In Fig.6 two characteristic curves were presented, for two values of maximum swing angle, 20^o and 40^o. For the given range of rotational speed, the dynamic pressure increase in the CRF related to the maximum value for the given α_0 is depicted. For $\alpha_0 = 20^o$ the maximum value of dynamic pressure was equal to 8 Pa, whereas for $\alpha_0 = 40^o$ was almost 100% higher.

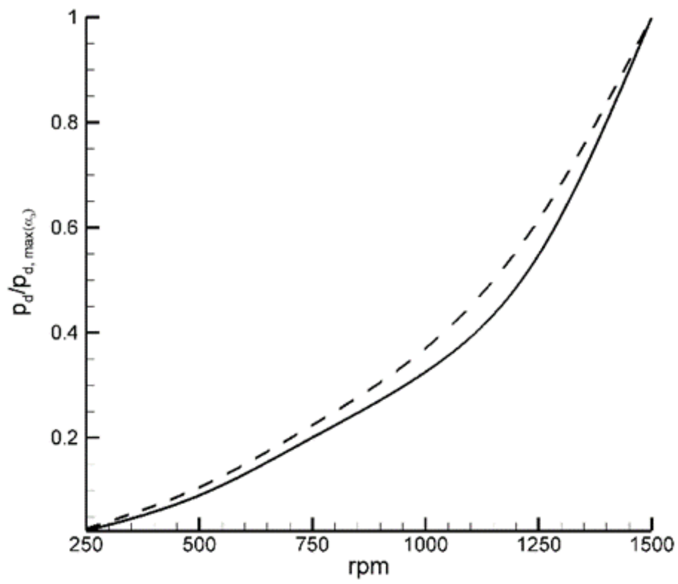


Figure 6 Performance characteristic of the cycloidal rotor fan (- - - $\alpha_0 = 20^o$ —— $\alpha_0 = 40^o$

One can see that for both α_0 values the dynamic pressure rises rapidly with the rotational speed, however, for higher α_0 this increase is slightly more rapid.

5. Summary and conclusion

The presented cross-flow fan with cycloidal rotor seems to be a promising alternative for applications where the open rotor design is required. A typical cross-flow fan with diffuser has main limitations concerned with flow direction, which is not easily changeable. Whereas, the cycloidal rotor fan is able to change the air flow direction in easy way, even during the operation. CRFs can be used in many industrial application from HVAC, through pneumatic conveying (e.g. ash) to cooling systems (e.g. in metallurgy).

Presented performance characteristic, which is similar for numerical and experimental approach, shows a promising potential of proposed CRF solution. However, there is still a lot of parameters which may affect the improvement of this characteristic, such as:

1. blades airfoil,
2. the maximum value of blade swing angle, α_0,
3. blade number
4. and blade chord-to-radius ratio.

According to the literature survey (see e.g. [3]), the optimization of the proposed design for obtaining better CRF performance is necessary and it is planned in the future. To this end, the prepared CFD model will be employed together with experimental investigation of the optimized design.

6. Acknowledgements

The presented results were performed within Statutory Research Funds of the Institute of Power Engineering and Turbomachinery of the Silesian University of Technology and within the project UMO-2014/15/B/ST8/00203 of National Science Centre.

References

[1] **Hwang, S., Min, Y., Lee, H., and Kim, J.**: Development of a Four-Rotor Cyclocopter, *Journal of Aircraft*, 45, 6, 2151–2157, **2008**.

[2] **Xisto, C. M., Pascoa, J. C., Abdollahzadeh, M., Leger, J. A., Schwaiger, M., and Wills, D.**: Pecyt - plasma enhanced cycloidal thruster, *50th AIAA/ASME/SAE/ASEE Joint Propulsion Conference*, **2014**.

[3] **Xisto, C. , Leger, J. , Páscoa, J. et al.**: Parametric Analysis of a Large-Scale Cycloidal Rotor in Hovering Conditions, *Journal of Aerospace Engineering*, 30, 1, **2016**.

[4] **Gabi, M., Klemm, T.**: Numerical and experimental investigations of Cross-flow fans, *Journal of Computational and Applied Mechanics*, 5, 2, 251–261, **2004**.

[5] **Witkowski, A., Majkut, M.**: Experimental investigation of inlet guide vane-rotor interaction in a low speed axial flow compressor stage, *Archive of Mechanical Engineering*, LIII, 1, 23–48, **2006**.