Aerodynamic assessment of the cycloidal rotor fan

S Dykas¹, M Majkut¹, K Smolka¹, M Strozik¹, T Staško¹
¹ Silesian University of Technology, Institute of Power Engineering and Turbomachinery, Konarskiego 18, 44-100 Gliwice, Poland

E-mail: Krystian.Smolka@polsl.pl

Abstract This paper presents numerical and experimental testing of a cycloidal rotor fan (CRF). The tested cyclorotor is made of four NACA0012 profile blades. CFD calculations using the Ansys CFX® tool and the LDA measurement technique are performed to determine the fan performance for different operating parameters, i.e. rotational speed and the rotor blade maximum swing angle values. One initial position of the blades corresponding to one air-flow direction is considered only. The comparison between the numerical results and the experimental LDA measurements is satisfactory and encouraging further investigation.

Symbols
\( \alpha \) – blade actual swing angle
\( \alpha_0 \) – blade total swing angle
\( c \) – blade chord, m
\( \Delta t \) – time step, s
\( \gamma \) – angle of the blade initial position
\( R \) – rotor radius, m
\( t \) – time, s
\( v \) – velocity, m/s
\( \omega \) – angular velocity, rad·s\(^{-1}\)

1. Introduction
The cycloidal rotor concept is mostly found in lift-based wind turbines (the Darrieus wind turbine) or in propulsion systems of unmanned aircrafts mainly [1-3]. This means that the solution can be applied to major types of turbomachines, such as engines (turbines) or working machines (compressors or pumps). The search for new applications and more and more efficient turbomachines, e.g. fans, encourages implementation of new ideas and designs. One of them is the idea of the cycloidal rotor fan (CRF) characterized by a wide range of rotational speeds and volume flow rates. This type of fan belongs to the family of cross-flow fans, but it has no casing (diffuser). Typical cross-flow fans (CFF’s) are widely used in cooling, heating and air conditioning systems, and in automotive as well as industrial applications, e.g. for heating and drying purposes. Most of them have a compact design and generate little noise. CFF’s are a particular type of fans having a drum-type rotor with a high span-to-radius ratio and a spiral-type casing. In CFF’s the flow field and the performance characteristics are more dependent on the casing shape than the rotor design, which is investigated both experimentally and numerically [4, 5].
The CRF is an open-rotor type fan and its main features are as follows: an arbitrary change in the flow direction without changing the fan position (only by changing the incident angle of the rotor blades), a big span of the rotor (making it possible to obtain a high volume flow rate or/and a big swept area), a uniform flow along the span (like in most cross-flow fans), a possibility of using the device with any gases (and even liquids, in the pump mode).

The solution can only be analysed through detailed numerical and experimental research. In the first step, a comparison was made between the results of a numerical model created in the Ansys CFX® program and the in-house experimental data. The good agreement between the two methods proves that a numerical model can be used to look for the most efficient solution of the CRF design.

A special script in the Ansys CFX Expression Language (CEL) had to be prepared to develop the numerical model of the air flow in the CRF. The presented experimental investigations draw on the Laser-Doppler Anemometry (LDA) (or the Laser Doppler Velocimetry (LDV)). The technique was developed over 40 years ago and is commonly applied by many researchers around the world in non-intrusive 1D, 2D and 3D point measurements of velocity and turbulence distribution in both free and internal flows. It was implemented in the Institute of Power Engineering and Turbomachinery and has been used there successfully for over 25 years in laboratories intended for the fluid-flow machinery testing, also to measure the flow field in compressors [6, 7].

2. Analysed object

The fan with a cycloidal rotor analysed herein is made of four NACA0012 profile blades with the chord of \( c = 50 \text{ mm} \). The NACA0012 profile is a fully symmetrical aerofoil with the thickness of 12\% of the chord length. The rotor radius totals \( R = 70 \text{ mm} \) and the initial position angles of the four blades under consideration are \( \gamma = 0, \pi/2, \pi \text{ and } 3\pi/2 \), respectively (cf. Fig. 1).

![Fig. 1. Configuration of the CRF with four NACA0012 blades](image-url)
around the point on its profile camber line in the middle of the chord. For a given rotational speed, the volume flow rate is controlled by the change in the blade total swing angle \((\alpha_0)\). The higher the blade total swing angle, the larger the volume flow should be.

3. Research methodology

3.1. CFD model

The numerical simulation of the unsteady flow in the cyclorotor fan was performed by means of the commercial Ansys CFX® software package. To this end, the whole computational domain consisted of four rotor-blade domains (one domain for one rotor blade) and one big domain surrounding the cyclorotor ensuring an undisturbed flow of the working medium (cf. Fig. 2). The “Opening” boundary condition is assumed for the main domain outer boundary, whereas the “General connection” boundary condition is assumed for interaction of the rotor-blade domains with the main domain. Compressible air is taken into account as the working medium. The k-\(\omega\) SST turbulence model is used in the computational model with a medium value of the turbulence level at the inlet. Because the CRF prototype was manufactured by means of 3D polymer-based printing methods, 0.1 mm sand-grain roughness is assumed for the blade walls in the numerical model.

![Computational domain and numerical mesh for the analysed CRF geometry](image)

The total number of the numerical mesh points is ~600k, whereas the main domain consists of ~400k points and each blade domain has ~50k mesh points (cf. Fig. 2). Because only three layers are used in the Z direction, the calculations are treated as two-dimensional.

In the Ansys CFX® software it was impossible to rotate the computational domains simultaneously around more than one coordinate system centre of rotation. For this reason, during the computations the rotor domains (domain 1, 2, 3 and 4 in Fig. 1) had to be moved (transformed) to the centre of the coordinate system, swung by angle \(\Delta\alpha\) and then transformed back to the original position and rotated by angle \(\omega t\). A CEL script had to be written for this purpose.

The transient calculations were performed with a constant time step calculated based on angular velocity:

\[
\Delta t = \frac{2\pi}{\omega n},
\]
where \( n \) is the constant used for keeping similar time step values for numerical simulations with different values of the cyclorotor angular speed \( \omega \).

Gradual increment in the blade swing angle in specific time steps was calculated according to the following relation:

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

It can be seen from relation (2) that \( \Delta \alpha \) may have a positive or a negative value, which ensures the blade return to the initial position after the rotor full rotation \((2\pi)\).

### 3.2. Experimental measurements

Additive manufacturing, which is more colloquially referred to as 3D printing, was used to make the cyclorotor fan (CRF) (cf. Fig. 3). All elements of the CRF were made of a polymer material by means of the FFF (fused filament fabrication) technique and roll bearings were used for all rotary connections. An electric motor with an inverter was used to drive the fan to obtain the required rotational speed.

The applied LDA system is a two-colour, six-beam, three-dimensional measuring system. It consists of a 4W argon-ion laser, tuned to 488 nm (blue) and 514.5 (green) lines, a transmitter, six beams, a three-dimensional measuring system with two 60 mm 1D and 2D laser probes. The focal length of the LDA probes is 160 mm or 400 mm with additional lenses producing the beam intersection lengths of about 0.66 mm or 4.09 mm, respectively. In order to obtain light impulses strong enough for the LDA, silicon oil particles with a mean diameter of 0.6 – 1.0 micrometre were used.

### 4. Results

Figures 4 and 5 show the velocity contours obtained from CFD calculations performed by means of the prepared model in the Ansys CFX® software (CRF anticlockwise rotation). The velocity contours are shown in two positions in time for one rotational speed value and for two values of the blade swing angle \( \alpha_0 \). It can be seen clearly that the flow velocity rises together with an increase in the blade total swing angle. This proves the ease of controlling the CRF performance: by increasing \( \alpha_0 \) it is possible to raise the volume flow rate at the same rotational speed.
The maximum velocity values are observed in the blade vicinity in the position angles between $\gamma = 0$ and $\gamma = \pi/2$. This is related to the non-optimal air flow around the blade for high positive values of the blade incident angle. The flow strong separation on the suction side and on the pressure side close to the trailing edge causes significant flow losses.

This phenomenon can be minimized by using other, more optimal profiles intended for operation at higher incident angle values. More symmetric aerofoils with a different thickness and location have to be tested for this kind of application. The effect can be also minimized by using optimal changes in the blade incident angle (blade swing angle $\alpha$), better adapted for and synchronized with the CRF angular velocity $\omega$. From the numerical point of view, it would be possible to do that by modifying relation (2), whereas technically, it would be very difficult to implement in a real machine. In such a case, the mechanical way of the incident angle adjustment would have to be replaced by electronic control drives, programmed individually for each blade.

Fig. 4 and Fig. 5 also indicate that for a smaller value of $\alpha_0$ there is a decrease in aerodynamic losses, which may suggest that to achieve a high volume flow rate, it is better to increase rotational speed rather than the blade total swing angle.

![Fig. 4. Velocity fields from CFD calculations with the SST turbulence model for 1000 rpm and $\alpha_0=20^\circ$ ($v = 0\div10$ m/s, $\Delta v = 1$ m/s)](image)

Fig. 6 presents the velocity profiles obtained from CFD calculations and from measurements. All the CFD results presented herein are averaged over one full rotation. The velocity profiles are presented at the distance of 140 mm (2R), 240 mm (4R) and 420 mm (6R) downstream from the CRF centre of rotation. The velocity profile obtained from experiments is presented for 31 measuring points with the pitch of 10 mm. The experimental results are presented at the distance of 140 mm (2R) only, which results from too big a measurement uncertainty in the case of distances 4R and 6R. A significant measurement uncertainty can also be observed for rotational speed lower than 500 rpm, especially for the lower value of $\alpha_0$.

Comparing the CFD and the experimental results, it may be concluded that the implemented Ansys CFX$^\text{®}$ model is able to predict the flow in the CRF with good accuracy in terms of the maximum value of velocity and velocity profiles.
Fig. 5 Velocity fields from CFD calculations with the SST turbulence model for 1000 rpm and $\alpha_0=40^\circ$ ($v = 0\text{--}10 \text{ m/s}$, $\Delta v = 1 \text{ m/s}$)

Fig. 6. Velocity profiles obtained from CFD calculations and measurements for 1000 rpm and two values of total swing angles
The observed differences between the results of CFD calculations and measurements necessitated more testing of the developed numerical model using other turbulence models. The calculations performed for most two-equation turbulence models gave results similar to calculations based on the k-\(\omega\) SST model. Only the results of calculations performed using the SAS turbulence model differ from the others. Fig. 7 presents a comparison between velocity profiles obtained using the SST and the SST-SAS turbulence models. The maximum value of velocity is similar for both numerical results, but there are small differences in the shape of the velocity profiles. The velocity profile for the results based on the SST-SAS turbulence model is less regular, but it can be expected that for a finer numerical mesh the results with the SAS turbulence model would be smoother. Figure 8 shows momentary velocity flow fields for the CFD results obtained using the SST-SAS turbulence model.

Considering the computational time and the lack of significant differences in the flow field generated by most turbulence models, it is recommended that the k-\(\omega\) SST turbulence model should be used for future calculations.

**Fig. 7.** Velocity profiles for CFD calculations with different turbulence models (1000 rpm, \(\alpha_0 = 40^\circ\))

**Fig. 8.** Velocity fields from CFD calculations with the SST-SAS turbulence model for 1000 rpm and \(\alpha_0 = 40^\circ\) (\(v = 0\pm 10\) m/s, \(\Delta v = 1\) m/s)
5. Summary and future plans
The paper presents an aerodynamic assessment of the cycloidal rotor fan based on CFD calculations and LDA flow field measurements. The numerical and experimental results are compared in terms of velocity profiles. Satisfactory agreement is shown between the numerical simulation results and the experimental data. The observed differences in the velocity flow field and the difficulties in the comparison for low rotational speed values between CFD calculations and measurements may be caused by:

- more significant than expected 3D effects in experimental measurements due to interaction between the side elements,
- small differences in swing angle values (α) between the numerical and the experimental model (the maximum difference was about 1°),
- considerable difficulties in the measurements using the LDA technique for low values of the flow velocity and high levels of turbulence.

Therefore, future works aiming to obtain a more efficient cyclorotor fan will be carried out using the CFD technique, and the final design will be tested further experimentally.

Future research will require numerical testing of the cyclorotor fan (CRF) design for different configurations of the:

- blade profile (e.g.: NACA0010, NACA0015, N66-012, E168, E169)
- blade chord (c),
- number of blades,
- blade swing centre of rotation,
- cyclorotor radius (R),
- blade initial swing angles (positions)
- value of the blade total swing angle (α0).

Acknowledgements
The presented results were obtained within Statutory Research Funds of the Institute of Power Engineering and Turbomachinery of the Silesian University of Technology.

References
[1] Hwang S, Min Y, Lee H, and Kim J 2008 Development of a Four-Rotor Cyclocopter Journal of Aircraft 45(6) pp. 2151-2157
[2] Xisto C M, Páscoa J C, Abdollahzadeh M, Leger J A, Schwaiger M and Wills D 2014 PECyT – plasma enhanced cycloidal thruster Proceedings of the 50th AIAA/ASME/SAE/ASEE Joint Propulsion Conference
[3] Xisto C M, Páscoa J C, Leger J A, Masarati P, Quaranta G, Morandini M, Gagnon L, Wills D and Schwaiger M 2014 Numerical modeling of geometrical effects in the performance of a cycloidal rotor Proceedings of the 6th European Conference on Computational Fluid Dynamics
[4] Gabi M, Dornstetter S and Klemm T 2002 Investigation of the flow field in crossflow fans by particle imaging velocimetry Proceedings of the 10th Int. Symp. On Flow Visualisation (Kyoto, Japan)
[5] Gabi M and Klemm T 2004 Numerical and experimental investigations of Cross-flow fans Journal of Computational and Applied Mechanics 5(2) pp. 251-261
[6] Witkowski A and Majkut M 2006 Experimental investigation of inlet guide vane-rotor interaction in a low-speed axial-flow compressor stage Archive of Mechanical Engineering LIII(1) pp. 23-48
[7] Witkowski A, Ziach M, Majkut M and Strozik M 2008 Experimental investigations of the flow phenomena in the rotor blades of the axial-flow low-speed compressor stage at the unstable part of the overall performance characteristic Archive of Mechanical Engineering LV(4) pp. 313-330