Methods of analyzing isolated and ducted fixed-pitch propellers under non-axial inflow conditions

M Cerny, J Faust, C Breitsamter
Technical University of Munich, Chair of Aerodynamics and Fluid Mechanics, Boltzmannstr. 15, 85748 Garching, Germany
E-mail: michael.cerny@tum.de

Abstract. Currently, a growing interest for the propulsion of unmanned aerial vehicles (UAVs) can be recognized. To address this, small-scale fixed-pitched propellers are analyzed in this work, both in isolated and ducted configurations. Due to their flight envelope, UAVs are likely to operate at highly inclined inflow conditions and even under reversed inflow, having a major influence on the flow field around a propeller. In order to investigate this influence, all analyses are performed at a range of inflow angles from $\alpha_{\text{disc}}=0^\circ$ to $180^\circ$ relative to the propeller axis. This paper employs different numerical techniques and compares them regarding their capability of showing the influence of non-axial inflow on propeller configurations. Wind tunnel experiments are performed measuring the forces and visualizing the flow field with PIV as well as steady and unsteady RANS simulations are applied to study details of the flow field.

1. Introduction
Propellers can be subjected to non-axial inflow conditions, due to a variety of circumstances, for example in very slow forward flight (like take-off and landing), during side-slip and during a turning flight [6]. Particularly in horizontal and edge-wise flight, conventional UAV multicopter configurations experience conditions which induce unsteady flow conditions [3, 14].

A large number of UAV designs apply ducted propellers since during a typical multicopter configuration mission, a significant amount of time will be spent in hover state, where a propeller mounted inside a duct is theoretically more efficient than an open propeller [15]. The flow characteristics of especially a ducted propeller under non-axial inflow conditions are reported less in the literature [9, 10, 11, 17]. In addition to this, the current work compares different analysis methods and their results to study the occurring flow.

Especially under non-axial inflow conditions, a propeller forms a highly complex three-dimensional flow field, resulting in comparison to an axially directed flow in additional unsteady forces and moments and a fluctuating behavior of the propeller thrust and torque. A detailed and illustrated explanation of the occurring flow effects responsible for this behavior has been given in [5]. In this work, the focus is set on the explanation and comparison of numerical methods used to study these non-axial inflow phenomena on the two exemplary configurations of an open and a ducted propeller configuration.

2. Experimental approach
The wind tunnel A of the Chair of Aerodynamics and Fluid Mechanics at the Technical University is used for performing all measurements. Its nozzle is 1.8 m in height, 2.4 m in...
width and its test section is 4.8 m long. The freestream turbulence intensity is 0.4 % at its maximum while the maximum velocity is 60 m/s.

Measurements are obtained for two-bladed fixed-pitch propellers APC 18x8E with a diameter of 18 inch (0.46 m) and a geometrical pitch of 8 inch giving a pitch angle at the radial location \( r/R = 0.75 \) of 10.7°. Fig. 1b shows their chord length and twist angle distribution.

The duct is designed for largely axial operating conditions, preferably with very low advance ratios. This leads to a large inlet radius and profile thickness and a wide diffusor angle. Fig. 1c shows the duct dimensions. If operated inside a duct, propellers are typically designed individually and show a long chord until the blade tip and blunt blade tips to reduce the flow through the blade tip gap [2]. In this work, to provide an easier comparison of both configurations, the same propeller is operated both as an open as well as a ducted propeller. For the latter the rounded blade tips are cut at a radius of \( r = 221 \text{ mm} \) which refers to a cut-off of about three percent.

![Figure 1](image1.png)

**Figure 1.** (a) Size, (b) twist angle and chord length distribution of the applied APC 18x8E propeller, (c) dimensions of the applied duct. [5]

The propeller is driven by a speed-controlled direct current motor (EC-60-167131, Maxon Motor). Its backside is connected to a six-component internal balance (K6D40 200N/5Nm/CG, ME-Messysteme GmbH) to obtain the aerodynamic loads with a resolution of 0.5 N / 0.005 Nm in thrust direction and 0.2 N / 0.005 Nm perpendicular to the thrust direction concerning the balance’s full range of measurements. Its sample rate is set to 300 Hz. To achieve steady loads the measurements are performed and averaged over thirty seconds (giving around 9000 samples). The assembly of motor and balance is surrounded by a 3D-printed closed self-holding fairing to measure the aerodynamic forces of the models only (see Fig. 2a). Between the motor and the
balance, a connector is integrated for the optional mounting of the duct. The whole system can be rotated in the wind tunnel, enabling the measurement at inflow angles from $\alpha_{\text{disc}} = 0^\circ$ to $180^\circ$ (see Fig. 2b).

The flow field is measured using stereo Particle Image Velocimetry (PIV) at a number of vertically oriented planes of measurement at the cross-section of the propeller. The PIV setup consists of the laser positioned above the wind tunnel on a traverse system. Two cameras are mounted on the side of the wind tunnel test section around one meter away of the flow section on another traverse system (see Fig. 2b). To detect the vortex structures in the wake, the PIV is performed phase-locked with the propeller’s rotation. Hereby, all measurements are performed with the propeller in a horizontally oriented position ($\zeta = 90^\circ$) to minimize laser shading. The PIV images read a spatial resolution of 3.3 mm. To obtain the mean velocity components, 200 image samples are time-averaged. Stanislas et al. state that with a proper calibration of the setup a precision of determining the displacements of around 0.1 to 0.3 pixels for each camera image can be obtained [12]. Therefore, the resulting uncertainty for the three velocity components can be quantified by two percent of the inflow velocity.

3. Free vortex method
A numerical low-fidelity solution methodology is with the use of a Lifting-Line model that is coupled to a free-wake model (see Fig. 3). The fundamental idea behind such approaches is to formulate the flow as two problems as suggested by [7]. The inner wing problem is associated with determining the bound circulation strength at each blade section, whilst the outer wake problem governs the advection of the vortex lattice trailing behind each rotor blade. The independent solutions are coupled through the induced velocity calculation at the blade collocation points. A detailed description of the method is given in [4].

4. Steady RANS simulations
A high-fidelity approach using a finite volume method solves the incompressible Reynolds Averaged Navier-Stokes (RANS) equations on a structured grid, meshed by the commercial software ANSYS ICEM CFD and consisting of four domains. One is an outer, static, hexahedral domain containing the wind tunnel support geometry and measuring an edge length, corresponding to approximately 30 times of the propeller diameter. Inside, a second cylindrical domain contains the propeller geometry for which the meshing is performed for the half of the cylinder, while the other side is mirrored. The obtained mesh sizes are listed in table 1.
Table 1. Mesh size (in million cells).

| Configuration | Outer domain | Prop. d. | Interface 1 d. | Interface 2 d. |
|---------------|--------------|----------|----------------|----------------|
| Isolated      | 11.72        | 4.9      | 0.54           | 0.54           |
| Ducted        | 11.99        | 5.22     | 0.7            | 0.66           |

Figure 4. (a) Blocking of the propeller domain, (b) meshing of the isolated configuration, (c) propeller mesh at $r/R = 0.7$, (d) meshing of the ducted configuration.

ANSYS Fluent is applied as flow solver. To calculate the flow field in a steady approach, the so-called Mixing Plane Model is used. The rotation of the propeller is realized by a transformation of the coordinate system of the inner domain. On the interfaces between the domains with a relative velocity to each other, Mixing Planes are applied. Hereby, the field variables are averaged and transferred to the adjacent domain as boundary conditions performed for every iteration of the calculation. [1]

Performing the averaging by an integration of the vectorial field values around one revolution, all vectors not perpendicular to the interface disc would be mathematically eliminated. To avoid this, the cylindrical interfaces are split in 16 circumferential segments with 40 radial ring segments $dA$ each, the mantle surfaces are divided into 12 segments (see Fig. 5a) with a separate Mixing Plane applied for every subarea. Doing so, the tangential vectorial values are preserved and transferred to the adjacent domain.
The segmentation of the propeller domain or the outer domain would affect their mesh quality in an undesirable manner since their blocking structure is set according to the contained geometry. Therefore, between both domains two additional thin interface domains are inserted and the segmentation is applied to them. The assembly of all four domains can be seen for both the open and the ducted propeller configuration in Fig. 5b.

Since the mesh of the propeller domain itself is not rotating but only its coordinate system, the influence of the propeller due to only one azimuthal position \( \zeta \) is accounted for. Hence, to consider different rotor positions, six simulations with the propeller azimuthal positions \( \zeta =\{0^\circ, 30^\circ, 60^\circ, 90^\circ, 120^\circ, 150^\circ\} \) are performed and the results averaged. Simulations with the propeller positions between \( \zeta = 180^\circ – 360^\circ \) are obsolete due to the symmetry of the propeller.

A pressure-based solver is applied using the SIMPLE algorithm. The fluid is modeled to be incompressible and the spatial gradients are discretized with a second order scheme. For the inlets, a velocity vector is defined, a turbulence level is prescribed to match the conditions of the wind tunnel experiments while the outlets are set to be pressure-outlets. The turbulence is modeled with the \( k-\omega \) shear stress transport (SST) model. Similar numerical settings have been employed previously, and were found to show close agreement with the experimental data [13, 16]. In all simulations, the rotor thrust was used as a convergence measure. The simulation duration largely depends on the inflow angle and unsteady aerodynamics involved, ranging from 200 to 3000 iterations to achieve a sufficiently converged solution.

5. Unsteady RANS simulations

The previously described RANS setup is adapted for unsteady calculations, transferring the mesh to an outer domain and an inner rotating domain. Between both, a sliding-mesh condition is applied. The chosen time step size of \( 4.167 \cdot 10^{-5} \) s is equivalent to a one degree azimuthal increment for the case of 4000 \( rpm \). The temporal gradients are discretized with a second order scheme. The other parameters are consistent with the steady state simulation setup. To achieve convergence, the simulations are performed over fifteen to twenty propeller revolutions, again depending on the inflow angle. The time-averaged loads are obtained by an averaging over the last revolution.

6. Results and Discussion

In a first step, the numerical methods described above are compared to the experimental results regarding their capability to predict the propeller thrust. The free wake method has been performed for the isolated configuration exclusively and shows results in good agreement with the experimental data for the whole inflow angle regime (see Fig. 6). The steady RANS simulations for the isolated propeller case are able to achieve satisfactory agreement with the experimental data at small inflow angles of \( \alpha_{disc} \leq 30^\circ \). For higher inflow angles, the thrust is
underpredicted. On the contrary, the thrust of the ducted configuration is met for inflow angles up to $\alpha_{disc} = 150^\circ$. The URANS calculations are in good agreement with the experimental data for both configurations up to $\alpha_{disc} = 150^\circ$.

![Graph showing thrust coefficient $c_T$ of open and ducted configurations](Image)

**Figure 6.** Thrust coefficient $c_T$ of the open and the ducted configuration as determined by different methods, $U_\infty = 10 \text{ m/s}$, $n = 4000 \text{ rpm}$.

The flow field of the cross-section of the propeller is illustrated by the induced axial velocity, normalized in a way to subtract the non-axial inflow component (Eq. 1). Further, the normalized vorticity perpendicular to the inflow is analyzed (Eq. 2)

$$U_{x,\text{norm}} = \frac{U_x}{U_\infty} - \cos \alpha_{disc}$$

$$\omega_{y,\text{norm}} = \frac{\omega_y}{\Omega}$$

with $U_x$ as the absolute velocity in x-direction (along the propeller axis), $\alpha_{disc}$ the inflow angle relative to the propeller plane and $\Omega$ denoting the propeller’s angular velocity.

![Images of propeller flow fields](Images)

**Figure 7.** Axial velocity and vorticity fields of the isolated propeller derived by (a,d) PIV measurements with blanked areas due to PIV laser sheet shading and reflections, (b,e) RANS calculations, and (c,f) URANS calculations, $U_\infty = 10 \text{ m/s}$, $\alpha_{disc} = 60^\circ$, $n = 4000 \text{ rpm}$. 


In Fig. 7, the axial velocity and vorticity fields of the isolated propeller are plotted as derived by PIV, the RANS and the URANS calculations for a selected operating condition. The URANS calculations are in good agreement with the experimental data showing both the flow acceleration as well as the trace of the blade tip vortices and the blade wake precisely. Contrary, the RANS mixing plane simulations show significant deviations in comparison to the experimental data. The blade tips cannot be predicted by the steady state approach, but the flow acceleration is underpredicted as well, correlated with the underpredicted propeller thrust.

Similar to the propeller thrust, the flow field calculations of the RANS simulations are much closer to the experiment for the ducted configuration (see Fig. 8). Driving flow effects, like the recirculation zones at the leeward duct side and inside the duct at the windward side as well as the propeller wake acceleration are well represented. Since characteristic vortex structures like the blade tip vortices or the blade wakes are much less distinct with the ducted configuration, less information are lost by using the steady state approach. Additionally, the flow through the duct throat results in a flow deflection with a more perpendicular orientation of the velocity vectors relative to the Mixing Plane. It seems that both effects are beneficial for the Mixing Plane method, which is originally intended for the calculation of compressor or turbine stages with an interal and primarily axially oriented flow. The steady state solution was found to remain attached at the duct’s inside leeward wall in response to a strong axial mass flow. This aspect does not appear physical, and the separation is most likely caused by the swirl velocity in the wake that cannot be accounted for using the RANS approach. Furthermore, the flow separation downstream of the wind tunnel support is not met resulting in a propeller wake which is oriented too strong along the propeller axis.

![Figure 8](image_url)

**Figure 8.** Axial velocity and vorticity fields of the ducted propeller derived by (a,d) PIV measurements with blanked areas due to PIV laser sheet shading and reflections, (b,e) RANS calculations, and (c,f) URANS calculations, \( U_\infty = 10 \text{ m/s}, \alpha_{disc} = 30^\circ, n = 4000 \text{ rpm}. \)

The URANS calculations show results which are in good agreement with the experimental data, including unsteady effects like the blade wakes (BW). The flow acceleration and the
formation of shear layers (ShrL) and the root vortex (RV) meet the experimental characteristics as well as the lateral deflection of the propeller wake. Consequently, the URANS results capture all relevant flow field effects with sufficient accuracy.

7. Conclusion

Three different numerical methods have been analyzed regarding their capability to predict the thrust and the flow field of an isolated and a ducted propeller at severe inclined inflow. The results indicate, that the free wake method is a very cost-effective and precise tool to predict the propeller thrust even under very high inflow angles. The Mixing Plane method can predict the flow field for limited inflow conditions only. For open propellers, especially under non-axial inflow, the method fails to predict thrust and flow field effects of acceptable accuracy. For an analysis of flow field characteristics and the interaction phenomena of different components URANS calculations are recommended.

Acknowledgments

This research was funded by the Bavarian Research Foundation (Bayerische Forschungsstiftung) under the grant number AZ-1277-17. Furthermore, the successful collaboration with the project partner Rolls-Royce Deutschland is highly acknowledged. Special thanks are addressed to ANSYS for providing the flow simulation software.

References

[1] ANSYS Inc. 2009 ANSYS Fluent: Theory Guide
[2] Bontempo R, M Cardone, M Manna and G T Vorraro 2014: Ducted Propeller Flow Analysis by Means of a Generalized Actuator Disk Model Energy Procedia vol 45 p 1107–1115
[3] Bristeau P J, P Martin, E Salan and N Petit 2009: The Role of Propeller Aerodynamics in the Model of a Quadrotor UAV European Control Conference (Budapest, Hungary) p 683–688
[4] Cerny M, N Herzog, J Faust, M Stuhlpfarrer and C Breitsamter 2018: Systematic Investigation of a Fixed-Pitch Small-Scale Propeller Under Non-Axial Inflow Conditions Deutscher Luft- und Raumfahrtkongress (DLRK, Friedrichshafen, Germany) no 642
[5] Cerny M and C Breitsamter 2020 Comparison of Isolated and Ducted Fixed-Pitch Propellers Under Non-Axial Inflow Conditions (Aerospace vol 7 no 112)
[6] Glauert H 1935 Airplane Propellers, Aerodynamic Theory - Division L ed W F Durand (Berlin, Germany: Julius Springer) p 32–58
[7] Johnson W 2017 CAMRAD II Comprehensive Analytical Model of Rotorcraft Aerodynamics and Dynamics Volume I: Theory (Palo Alto, California: Johnson Aeronautics)
[8] Lindenburg C 2004: Modelling of Rotational Augmentation Based on Engineering Considerations and Measurements European Wind Energy Conference (London, UK)
[9] Pereira J L 2008 Hover and Wind-Tunnel Testing of Shrouded Rotors for Improved Micro Air Vehicle Design (College Park: University of Maryland)
[10] Russell C R and M K Sekula 2017: Comprehensive Analysis Modeling of Small-Scale UAS Rotors AHS International 73rd Annual Forum & Technology Display (Fort Worth, Texas, USA)
[11] Russell C R, C R Theodore and M K Sekula 2018: Incorporating Test Data for Small UAS at the Conceptual Design Level AHS International Technical Conference on Aeromechanics Design for Transformative Vertical Flight (San Francisco, California, USA)
[12] Stanislas M, K Okamoto C J Kühler and J Westerweel 2005 Main results of the second international PIV challenge (Exp. Fluids vol 39 no 2) p 170–191
[13] Stuhlpfarrer M, A Valero-Andreu and C Breitsamter 2017 Numerical and experimental investigations of the propeller characteristics of an electrically powered ultralight aircraft (CEAS Aeron. J vol. 8 no 3) p 441–460
[14] Theys B et al 2014: Wind Tunnel Testing of a VTOL MAV Propeller in Tilted Operating Mode International Conference on Unmanned Aircraft Systems (ICUAS) (Orlando, Florida, USA) p 1064–1072
[15] Weir R J 1987 Ducted Propeller Design and Analysis (Sandia Report SAND87-211, UC-32)
[16] You J H, C Breitsamter and R Heger 2016 Numerical investigations of Fenestron noise characteristics using a hybrid method (CEAS Aeronautical Journal vol 7 no 2) p 185–207
[17] Zhou W, Z Ning, H Li and H Hu 2017: An Experimental Investigation on Rotor-to-Rotor Interactions of Small UAV Propellers 35th AIAA Applied Aerodynamics Conference (Denver, Colorado, USA)