9th International Congress of the Serbian Society of Mechanics
July 5-7, 2023, Vrnjačka Banja, Serbia

NUMERICAL INVESTIGATION OF FLOWS AROUND SMALL-SCALE PROPELLERS: POSSIBILITIES AND CHALLENGES

Jelena Svorcan

Department of Aerospace Engineering, Faculty of Mechanical Engineering, University of Belgrade, Kraljice Marije 16, 11120 Belgrade 35
e-mail: jsvorcan@mas.bg.ac.rs

Abstract

This paper focuses on flows induced by small-scale propeller blades and the wakes shedding from their tips. Flows around propellers for unmanned air vehicles (approximately 25-75 cm in diameter) in hover are simulated by different approaches to considering turbulence. The challenges to simulating these kinds of flows mainly arise from the relatively low values of Reynolds numbers (several tens to several hundreds of thousands) when transition and other flow phenomena may be expected. The adopted numerical set-ups are validated though comparisons with available experimental data. It can be concluded that global aerodynamic performance can be determined with satisfactory accuracy (the discrepancies between computed and measured values of thrust and torque remain below several percents). However, discerning the actual flow characteristics remains challenging. Here, some distinguishing features of small Re rotational flows are accentuated and discussed. Vortex wakes shedding from the blades are visualized and analyzed. These two benchmark examples provide useful guidelines for further numerical and experimental studies of small-scale propellers.

Key words: UAV propeller, aerodynamic performance, hover, turbulence, wake.

1. Introduction

Modern day small-scale rotors common for unmanned aircraft or future urban air vehicles should satisfy several requirements: increased aerodynamic performance, reliability in various operating regimes, low noise. To satisfy these goals, engineers should first comprehend the induced flow fields in hover. For that reason, both numerical and experimental investigations of small-scale rotors have been intensifying over the past years [1-7]. This research also builds on previous studies [8,9] and focuses on low-to-medium Reynolds number (Re) flows forming around two different small-scale propeller rotors in hover.

Distributions of aerodynamic loads along rotating blades influence the two fundamental global aerodynamic quantifiers of propellers: generated thrust T at a required torque Q (power P), that are necessary for subsequent analyses (such as flight dynamics, stability and control, structural reliability) and determine the success of the complete system/aircraft. Unfortunately, there are numerous challenges to their accurate determination, both experimental and computational. On one hand, blade dimensions are small and real geometric features deviate from the modeled, airfoils designed for low Re are somewhat specific, insufficiently tested and highly sensitive to outer disturbances, whilst high angular velocities require the use of sophisticated measuring equipment. Even when great care is taken, error bars appear quite wide, particularly when torque is investigated [6]. On the other hand, to provide reliable and usable results, contemporary numerical simulations require refined meshes, miniscule time scales and advanced approach to modeling/partially resolving turbulence [9,10].

In order to assess the possibilities and benefits of different numerical approaches, flows around two hovering small-scale propellers (approximately 25 cm and 75 cm in diameter) are computed and analyzed in more detail. Different flow assumptions befitting rotational lifting surfaces are employed, ranging from the simplest (BEMT, vortex models) to more complex, such as RANS and ultimately wall-modelled large eddy simulation (WMLES).

2. Formulation of the numerical models

Propeller analysis usually starts with hover, the most basic flight condition. Just how challenging this topic is, can be proved by the fact that AIAA Rotorcraft Hover Prediction Workshop is held every year at AIAA SciTech international conference (https://www.aiaa-hpw.org/hpw-vision). Challenges to both simulation and experimental measurements include zero inflow across outer boundaries, induced velocity field around the rotor, accurate load prediction, transition to turbulence, wake/tip vortex formation, expansion, development, interaction with the blades and final breakdown, aeroelasticity of the blades, appropriate visualization of the flow field, unconventional configurations (mutual effects of multiple rotors), performance of channeled rotors, aerodynamic noise, etc. To deal with these challenges, one (or all) of the following three categories of computational models is usually employed.

2.1 BEMT

Blade Element Momentum Theory (BEMT), derived from combining MT and BET, is a very simple but surprisingly accurate computational model, that is still very much employed [7,11]. Its major advantages are simplicity, ease of implementation, speed, and usable results. Some of the main disadvantages are that it assumes stationary, inviscid flow (which can be implicitly mitigated to some extent) and requires the knowledge of airfoil aerodynamic characteristics (which is often not available, particularly for novel/unconventional airfoils). The main outcomes from the computation are total thrust and torque (as well as the distributions of their derivatives along the blade) resulting from the complete velocity triangles on blade segments that are known once the induced velocity field is iteratively determined.

2.2 Vortex models

Whereas many variants exist (a very nice review is provided in [12]), vortex methods assume potential and most often unsteady flow. The leading result is a truthful representation of the wake that enables the analysis of its shape and effects on surrounding objects/structures. However, given that the computational cost is not negligible, computational fluid dynamics (CFD) techniques are much more common today.

2.3 CFD

Here, we also encounter many different approaches. The most employed are still the (unsteady) Reynolds-averaged Navier-Stokes, (U)RANS, equations [7,8], but more complex models, that resolve at least a portion of the turbulence spectrum (like DES or LES) are also being tested [9,10]. The primary advantage of RANS approach is computational simplicity. By assuming a quasi-steady flow field where inertial terms from the rotation are added to the equations, it is possible to obtain a reasonable preliminary estimation of averaged aerodynamic loads. In the second phase, it is also possible to simulate the rotor rotation by actually moving a part of the mesh, which enables the consideration of transient effects. On the other hand, LES and DES are computationally still very costly since they require small spatial and temporal scales. They perform spatial filtering, which in turn, requires adequate sub-grid scale (SGS) modelling as well as special treatment of flow adjacent to the walls. In wall-modelled large eddy simulation (WMLES) larger-scale turbulent motion is resolved, while subgrid-scale motion (appearing in the wall vicinity) is considered more isotropic and can be modelled [13].

The choice of the numerical model dictates the way the geometry and computational mesh are generated. Usually, the domain is in a shape of a revolution body, whereas grids are hybrid unstructured, refined in the wall vicinity (to capture the sudden changes of flow quantities inside the boundary layer) and aft of the rotor to better capture the tip vortices and the developing wake. It is very important to perform valid grid convergence studies since numerical dissipation and instabilities are common. Adequate representation of the blade leading edge is also imperative since it directly influences transition to turbulence and profile drag, particularly at low-to-medium Re (several tens to hundreds of thousands). Trip is often introduced to enforce the transition end ensure dominantly turbulent flow since transition may be quite hard to accurately simulate. One should either come very close to the wall (and drastically increase the mesh size) or use appropriate wall functions. Since transition to turbulence and the accompanying laminar separation bubble (LSB) are still unresolved flow phenomena, different wall functions are tried and used. Important contributors to drag and wake formation are blade root and tip segments so special attention should be paid to them as well. It is now known and confirmed that the shed wake is very complex, comprising both primary (helicoidal) and secondary structures (rings around the main helicoids) that evolve and break down [14]. Adaptive mesh refinement may be employed to capture the wake shape, but many rotor rotations are needed, in turn causing numerical instabilities [15]. For that reason, experimental validation is still very much desired and necessary.

The problem becomes even more complicated if we take a step from the idealized, isolated, and rigid rotor to the installed elastic rotor (in full configuration) or in ground effect. Even the comparison to experimental measurements is not straightforward since the experimental results are affected by the fuselage, test stand and measuring equipment or wind tunnel walls and should be corrected (which introduces additional uncertainties to the analysis).

Possible outcomes from CFD analyses are extensive, both quantitative (thrust and torque/power, pressure and wall shear stress distributions) and qualitative (different visualizations of the flow field such as pressure or velocity contours, velocity vectors, wake shape, etc.).

3. Problem description

The geometric shape of small-scale propeller blades is usually quite complex and curved. It is defined by the spanwise distributions of airfoil (thickness and curvature), pitch and chord, which may be dominantly non-linear due to the use of modern materials and manufacturing technologies. Here, two different rotors (approximately 25 cm and 75 cm in diameter) are considered. Their detailed geometric descriptions may be found in [6,7].

Computations are performed using the finite-volume-based flow solvers charLES (compressible) and ANSYS FLUENT (incompressible). Fine meshes are generated to achieve fine spatial length scales. Small time steps and higher order discretization schemes are used. Zero-gauge pressures are assumed at the outer boundaries. To simulate rotation, computational domains are split into two zones where the inner zones, encompassing the propellers, rotate.

4. Results and discussion

Firstly, computed thrust and torque/power values are compared to the available experimental data [6,7]. As illustrated in Fig. 1, very satisfactory correspondence of thrust forces can be achieved even at Re spanning from approximately 80 to 300 thousand. For both rotors, the slightly bigger (diameter 75 cm) on the left, and the smaller (diameter 25 cm) on the right, differences between numerical and measured values remain within the error bars.

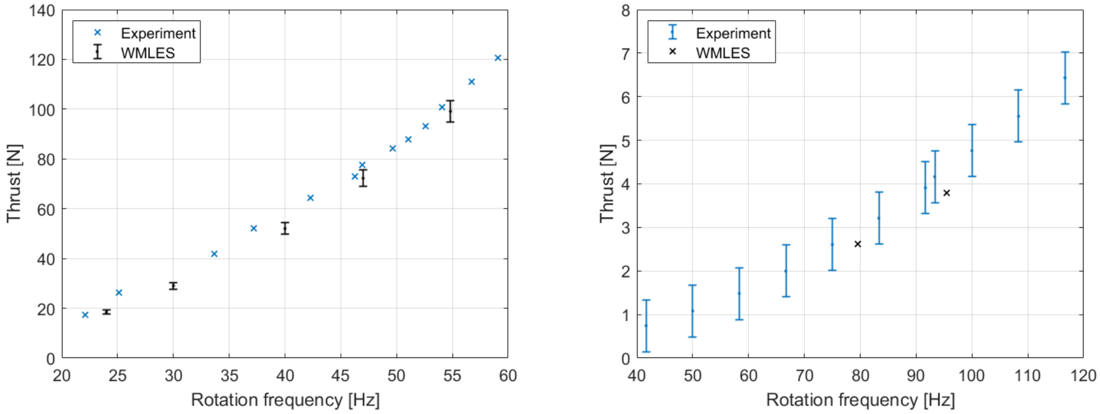

Fig. 1. Computed vs. measured thrust force: the 75 cm diameter propeller (left), the 25 cm diameter propeller (right).

The computed vortices, coloured by velocity, shedding from the blade tips of the 25 cm diameter propeller, viewed from two different angles, are illustrated in Fig. 2. It can be observed that even the mesh of approximately 8 million control volumes may not be sufficient to accurately capture all the complexities of the wake that should comprise both primary helicoidal and secondary vortices (forming around the primary structures). However, primary structures as well as blade/wake interaction seem well apprehended. More representative flow visualizations, obtained on more refined meshes around the 75 cm diameter propeller and by the flow solver charLES, are available in [9].

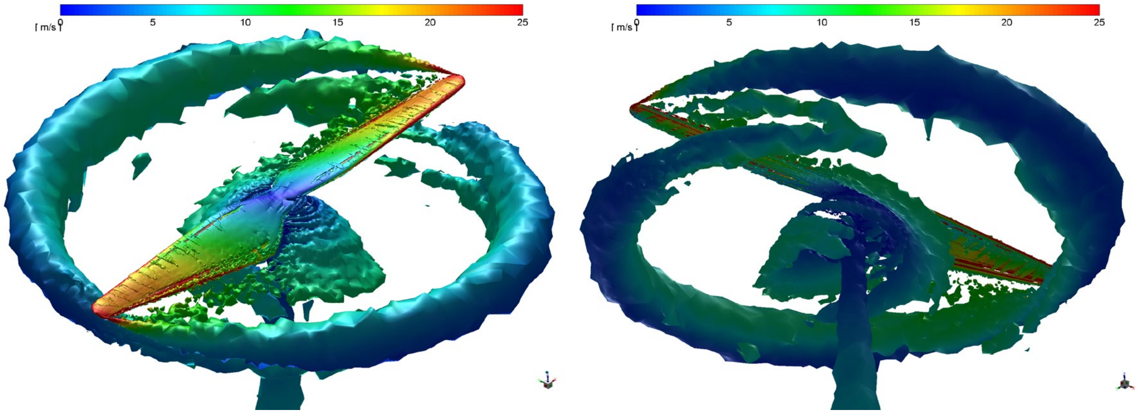

Fig. 2. Wake behind the 25 cm propeller computed by ANSYS FLUENT.

5. Conclusions

The paper addresses some of the issues present when simulating flows around propeller rotors. Some basic distinctions between large- and small-scale structures are outlined, in terms of geometry (small-scale ones are usually more curved), structural behavior (small-scale structures are less susceptible to aeroelastic effects), control algorithms (small-scale structures are frequency controlled), operating conditions (small-scale structures usually operate in “filthier” flows) and flow physics (small-scale structures usually involve both laminar and turbulent flows, as well as the computationally very delicate transitional flows).

The most employed computational techniques can roughly be divided into three main groups: BEMT, vortex, and CFD methods. Generally, BEMT is excellent for preliminary and optimization studies (estimated thrust error can be around 10%). Vortex methods are used when the wake is in focus, but may simplify the flow too much, and their computational cost is almost comparable to RANS, whereas CFD methods offer the most comprehensive results, but that will also depend on the starting assumptions (thrust errors may range from less than 1% to more than 10%). Ideally, experiment and LES should be used and combined whenever possible. In that respect, this study demonstrates that WMLES presents a valuable tool for the design and analysis of small-scale propeller blades. Additionally, some important flow features as well as the hovering propeller wake are discerned.

Current and future trends in the analysis of these flows move towards more complex simulations (hybrid DES and LES), stronger correlation to experimental measurements, further inclusion of AI methods, smart control, etc.

Acknowledgement: This research work is supported by the Ministry of Science, Technological Development and Innovation of the Republic of Serbia through contract no. 451-03-47/2023-01/200105.

References:

[1] Carreño Ruiz M., Scanavino M., D'Ambrosio D., Guglieri G., Vilardi A., *Experimental and numerical analysis of hovering multicopter performance in low-Reynolds number conditions*, Aerospace Science and Technology, Vol. 128, 107777, 2022.

[2] Wang C., Li P., Guo C., Wang L., Sun S., *Numerical research on the instabilities of CLT propeller wake*, Ocean Engineering, Vol. 243, 110305, 2022.

[3] Zhang T., Barakos G.N., *High-fidelity numerical analysis and optimisation of ducted propeller aerodynamics and acoustics*, Aerospace Science and Technology, Vol. 113, 106708, 2021.

[4] Garofano-Soldado A., Sanchez-Cuevas P.J., Heredia G., Ollero A., *Numerical-experimental evaluation and modelling of aerodynamic ground effect for small-scale tilted propellers at low Reynolds numbers*, Aerospace Science and Technology, Vol. 126, 107625, 2022.

[5] Nathanael J.C., Wang C.-H.J., Low K.H., *Numerical studies on modeling the near- and far-field wake vortex of a quadrotor in forward flight*, Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, Vol. 236(6), 1166-1183, 2022.

[6] Russell C., Willink G., Theodore C., Jung J., Glasner B., *Wind tunnel and hover performance test results for multicopter UAS vehicles*, NASA/TM−2018−219758, NASA Ames Research Center, Moffett Field, CA, 2018.

[7] Kovačević A., Svorcan J., Hasan M.S., Ivanov T., Jovanović M., *Optimal propeller blade design, computation, manufacturing and experimental testing*, Aircraft Engineering and Aerospace Technology, Vol. 93(8), 1323-1332, 2021.

[8] Svorcan J., Kovačević A., Tanović D., Hasan M.S., *Towards viable flow simulations of small-scale rotors and blade segments*, Theoretical and Applied Mechanics, Vol. 48(2), 143-157, 2021.

[9] Svorcan J., Wang K., Ivey C., Kovačević A., *Estimating the performance of a small-scale non-isolated propeller in hover using WMLES*, Center for Turbulence Research Annual Research Briefs, 87-95, 2022.

[10] Stratton Z., Spyropoulos J.T., Bose S., Svorcan J., *Rotor performance and turbulent wake simulations of a scaled helicopter rotor in hover using wall-modeled large-eddy simulations*, AIAA SciTech Forum, AIAA 2023-2636, 2023.
[11] Svorcan J., Hasan M.S., Kovačević A., Ivanov T., *Design of HALE propeller through multi-objective optimization*, AIAA Propulsion and Energy Forum, AIAA 2021-3730, 2021.
[12] Lee H., Sengupta B., Araghizadeh M.S., Myong R.S., *Review of vortex methods for rotor aerodynamics and wake dynamics*, Advances in Aerodynamics, Vol. 4(1), 20, 2022.
[13] Moin P., Bodart J., Bose S., Park G.I., *Wall-modeling in complex turbulent flows*, Notes on Numerical Fluid Mechanics and Multidisciplinary Design, Vol. 133, 207-219, 2016.
[14] Schwarz C., Bodling A., Wolf C.C., Brinkema R., Potsdam M., Gardner A.D., *Development of secondary vortex structures in rotor wakes*, Experiments in Fluids, Vol. 63(1), 4, 2022.
[15] Abras J.N., Narducci R., Hariharan N., *Impact of high-fidelity simulation variations on wake breakdown of a rotor in hover*, AIAA SciTech 2020 Forum, AIAA 2020-0531, 2020.